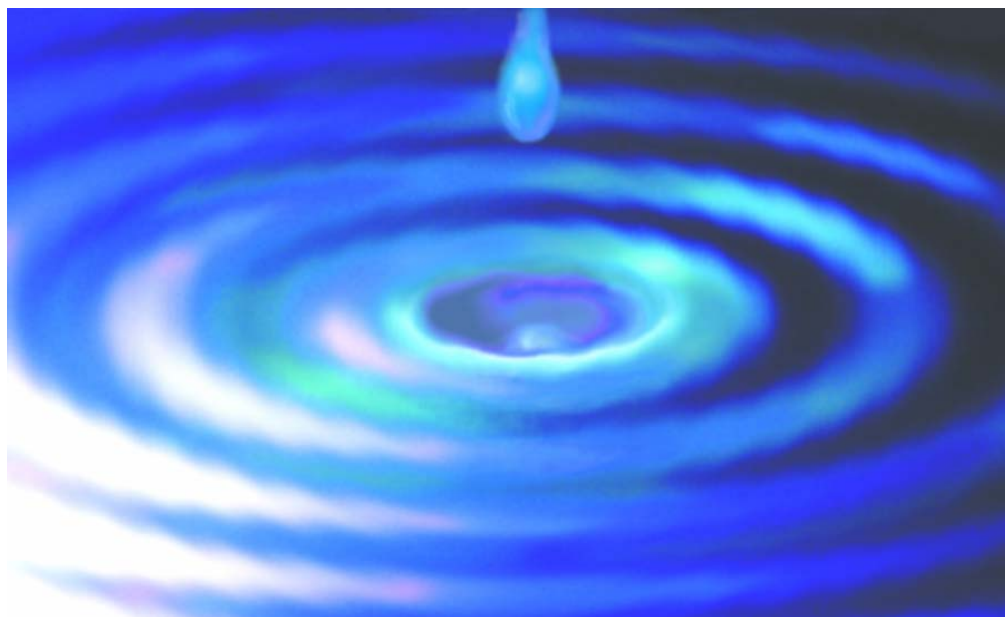


SMS



TUTORIALS

Version 9.0

SMS 9.0

Copyright © 2005 Brigham Young University – Environmental Modeling Research Laboratory August 12, 2005.

All Rights Reserved

Unauthorized duplication of the *SMS* software or user's manual is strictly prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENVIRONMENTAL MODELING RESEARCH LABORATORY MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *SMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS MANUAL

The software *SMS* is a product of the Environmental Modeling Research Laboratory of Brigham Young University

www.emrl.byu.edu

Last Revision: August 12, 2005

Introduction

This document contains tutorials for the Surface-water Modeling System (SMS) version 9.0. Each tutorial is meant to provide training on a specific component of SMS. It is strongly suggested that you complete the applicable tutorials before using SMS on a routine basis. For additional training, contact your SMS distributor.

SMS is a pre- and post-processor for surface water modeling and analysis. It includes one-, two- and three-dimensional numeric models including lumped parameters (step backwater), finite element and finite difference models. Interfaces specifically designed to facilitate the utilization of several numerical models comprise the modules of SMS. Supported models include:

- Two-dimensional riverine/estuarine circulation models *RMA2*, *HIVEL2D* and *FESWMS*.
- Three-dimensional riverine/estuarine circulation models *RMA10* and *CH3D*.
- Ocean circulation models *ADCIRC*, *M2D* and *BOUSS2D*.
- Phase resolving wave models *CGWAVE* and *BOUSS2D*.
- Non-phase resolving wave model *STWAVE*.
- Transport models *RMA4* and *SED2D-WES*.
- One-dimensional riverine model *HEC-RAS*.
- Lagrangian particle tracking model *PTM*.

The interfaces in *SMS* are endorsed by the USACE-Engineering Research and Development Center (ERDC) at the Waterways Experiment Station (WES) as well as the Federal Highway Administration (FHWA).

Each numerical model is designed to address a specific class of problems. Some calculate hydrodynamic data such as water surface elevations and flow velocities. Others compute wave mechanics such as wave height and direction. Still others track contaminant migration or suspended sediment concentrations. Some of the models support both steady-state and unsteady (dynamic) analyses, while others support only steady-state analysis. Some support supercritical flow, while others support only subcritical.

The finite element mesh, finite difference grid or cross section entities, along with associated boundary conditions necessary for analysis, are created within *SMS* and then saved to model-specific files. These files are used as input to the hydrodynamic, wave mechanic, contaminant migration and sediment transport analysis engines. The numerical models create solution files that contain the water surface elevations, flow velocities, contaminant concentrations, sediment concentrations or other functional data at each node, cell or section. *SMS* reads this data to create plots and animations.

SMS can also be used as a pre- and post-processor for other finite element or finite difference programs as long as the programs can read and write files in a supported format. To facilitate this, a *generic* interface is available to define parameters for a proprietary model. *SMS* is well suited for the construction of large, complex meshes (up to hundreds of thousands of elements) of arbitrary shape.

Please note that in these tutorials, reference to a menu item will be as follows: *Menu | Menu-Item*. For example: *File | Exit* indicates to select the *Exit* item from the *File* menu.

1.1 SMS Help ---

Accompanying *SMS* is the *SMS online Help*, which fully describes the available options in each dialog box. The help can be accessed through the *Help* menu inside of *SMS* or from the *Help* button on each dialog box. In addition, help files are available for some of the numerical models supported by *SMS*.

1.2 Suggested Order of Completion ---

Most of these lessons are developed for two dimensional finite element meshes and finite difference grids. If you want to use *SMS* for its *River Hydraulic Module* and *HECRAS* interface, you may want to examine the Lesson 2 and then skip to Lesson 22.

Users interested in finite element modeling should review Lessons 2 through 5. These lessons describe methods for generating finite element meshes, visualizing data on

these meshes, and extracting data from spatially varied data. They also illustrate how to evaluate model sensitivity and perform model verification. The techniques are illustrated using a riverine finite element model, but the tools apply equally as well to coastal applications and finite difference applications. For this reason, finite difference modelers should probably review these lessons as well.

Once the user understands the basic layout of *SMS*, he/she should proceed to the lessons which illustrate the specific model that will be used.

Each lesson includes both input and output files to facilitate rapid evaluation of the model and the capabilities.

1.3 Demo vs. Working Modes

Most users do not require all modules or model interfaces provided in *SMS*; for this reason, the interface is partitioned to allow the user to work only with data applicable to his/her study. Each module or model interface can be licensed individually. When running in regular mode, *SMS* disables unlicensed features. It is possible, however, to access all modules and model interfaces in *SMS* by running in *Demo Mode*.

If you have not licensed any part of *SMS*, it will automatically run in *Demo Mode*. On the other hand, if you do have a license for part of *SMS* and would like to experiment with an unlicensed module, you can tell *SMS* to run in *Demo Mode*. To do this:

1. Select *File | Demo Mode*.
2. Select *Yes* to the prompt, *Are you sure you want to delete everything?*

When you do this, a check mark appears next to this menu command. *Demo Mode* in *SMS* allows access to all functions except the *Save* and *Print* commands. To get back to normal operation mode, select this menu item a second time – you will once again be required to delete all data. Note that if you have no registered modules, you cannot leave *Demo Mode*.

Overview of SMS

2.1 Introduction

This tutorial describes the major components of the *SMS* interface and gives a brief introduction to the different *SMS* modules. It is suggested that this tutorial be completed before any other tutorial. All files for this tutorial are in the *tutorial\tut02* directory.

2.2 Getting Started

Before beginning this tutorial you should have installed *SMS* on your computer. If you have not yet installed *SMS*, please do so before continuing. Each chapter of this tutorial document demonstrates the use of a specific component of *SMS*. If you have not purchased all modules of *SMS*, or if you are evaluating the software, you should run *SMS* in *Demo Mode* to complete this tutorial (see section 1.4). When using *Demo Mode*, you will not be able to save files. For this reason, all files that you are asked to save have been included in the *output* subdirectory under the *tutorial\tut02* directory. When you are asked to save a file, you should instead open the file from this *output* directory. To start *SMS*, do the following:

- Open the *Start* menu, scroll to *Programs*, select *SMS* and click on *SMS 9.0*.

2.3 The SMS Screen

The *SMS* screen is divided into six main sections: the *Main Graphics Window*, the *Project Explorer* (this may also be referred to as the *Tree Window*), the *Toolbars*, the *Edit Window*, the *Menu Bar* and the *Status Bars*, as shown in Figure 2-1. Normally the main graphics window fills the majority of the screen; however, plot windows can also be opened to display 2D plots of various data.

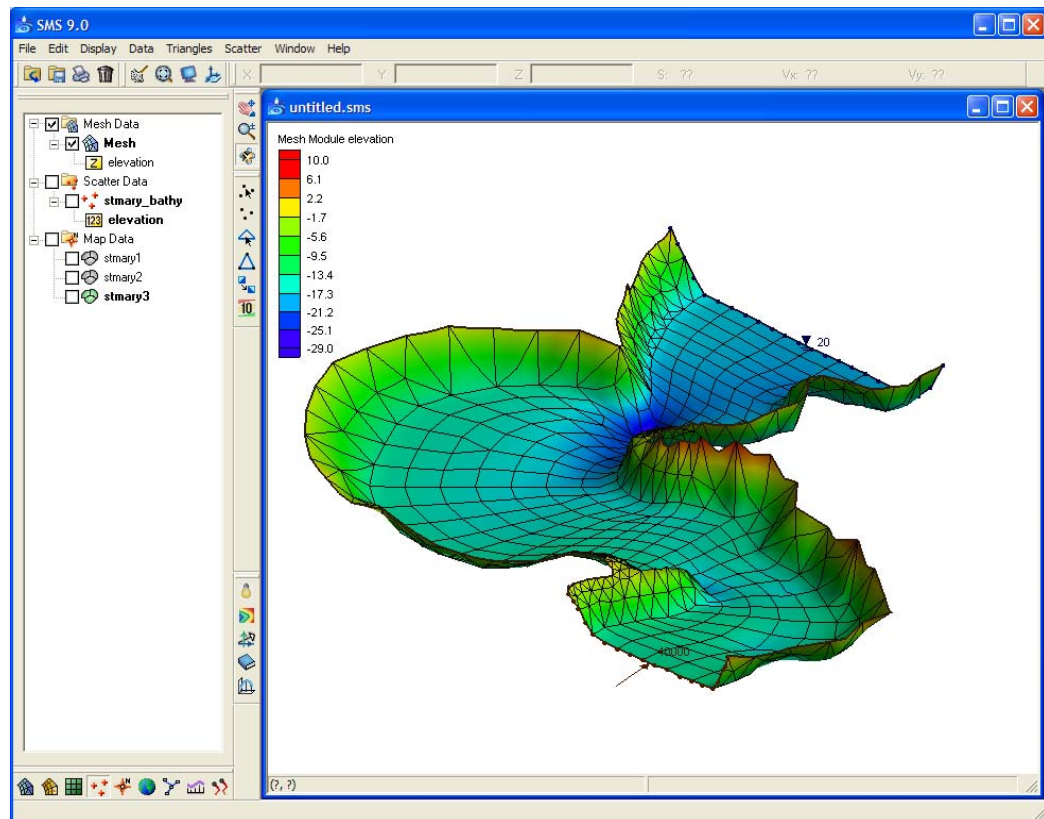


Figure 2-1. The SMS screen.

2.3.1 The Main Graphics Window

The *Main Graphics Window* is the biggest part of the *SMS* screen. Most of the data manipulation is done in this window. You will use it with each tutorial chapter.

2.3.2 The Toolbox

The *Toolbar* actually consists of multiple dockable toolbars. By default they are positioned at various locations on the left side application, but can be positioned around the interface as desired. The toolbars include:

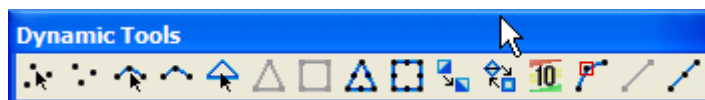
- **Modules.** This image shows the current *SMS Modules*. As described in the *SMS online Help* these icons control what menu commands and tools are available at any given time while operating in *SMS*. Each module corresponds to a specific type of data. For example, one icon corresponds to finite element meshes, one to Cartesian grids, and one to scattered data. If the scattered data module is active, the commands that operate on scattered data are available. The user can change modules by selecting the icon for the module, or selecting an entity in the *Project Explorer* or by right clicking in the *Project Explorer*. The module toolbar is displayed by default at the bottom of the application.



- **Static Tools.** This toolbar contains a set of tools that do not change for different modules. These tools are used for manipulating the display. By default they appear at the top of the display, between the *Project Explorer* and the *Graphics Window*.



- **Dynamic Tools.** These tools change according to the selected module and the active model. These tools are used for creating and editing entities specific to the module. They appear between the *Project Explorer* and the *Graphics Window* below the *Static Tools*.



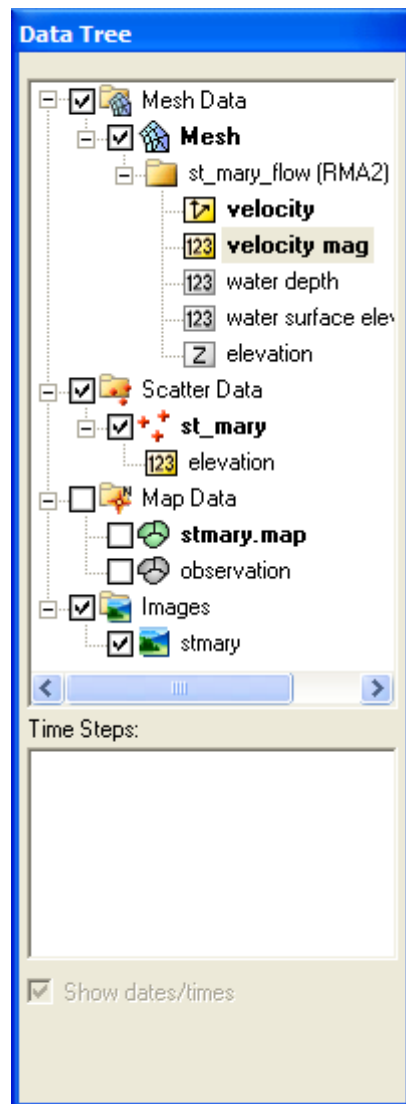
- **Macros.** There are three separate *Macro Toolbars*. These are shortcuts for menu commands. By default the standard macros and the file toolbar appear above the *Project Explorer* when displayed and the *Optional Macros* appear between the *Project Explorer* and the *Graphics Window* below the *Dynamic Tools*. Which macro toolbars appear at startup is set in the *Preferences Dialog* (Edit|Preferences command or right click in *Project Explorer* to set preferences).



2.3.3 The Project Explorer

The *Project Explorer Window* allows the user to view all the data that makes up a part of a project. It appears by default on the left side of the screen, but can be docked on either side, or viewed as a separate window.

It is used to switch modules, select a coverage to work with, select a data set and time step to be active, set display setting of the various entities the active coverage. By right clicking on various entities in the project explorer, the user may also transform, copy, or manipulate the entity.



2.3.4 The Edit Window

The *Edit Window* appears below the menus at the top of the application. It is used to show and/or change the coordinates of selected entities. It also displays the functional data for those selected entities.



2.3.5 The Menu Bar

The *Menu Bar* contains commands that are available for data manipulation. The menus shown in the *Menu Bar* depend on the active module and numerical model.

2.3.6 The Status Bars

There are two status bars: one at the bottom of the SMS application window and a second attached to the *Main Graphics Window*. The status bar attached to the main application window shows help messages when the mouse hovers over a tool or an item in a dialog box. At times, it also may display a message in red text to prompt for specific actions, such as that shown in the figure below.

A screenshot of a status bar with a blue border. It contains a red text message that reads: 'Enter the polygon by clicking in the window. Double-click to end.'

The second status bar, attached to the *Main Graphics Window*, is split into two separate panes. The left shows the mouse coordinates when the model is in plan view. The right pane shows information for selected entities.

A screenshot of a status bar with a blue border, split into two panes. The left pane displays mouse coordinates: '(3320.0, 4116.0, -15.09753065004) s: 0.5982480651959'. The right pane displays entity information: 'Node info: 1 selected; id = 1974; Attached to 4 elements.'

2.4 Using a Background Image

A good way to visualize the model is to import a digital image of the site. For this tutorial, an image was created by scanning a portion of a USGS quadrangle map and saving the scanned image as a *JPEG* file. SMS can open most common image formats including *TIFF*, *JPEG*, and *Mr.Sid* images. Once the image is inside SMS, it is displayed in plan view behind all other data, or it can be mapped as a texture onto a finite element mesh or triangulated scatter point surface.

2.4.1 Opening the Image

To open the *JPEG* image in this example:

1. Select *File | Open*.
2. Select the file *stmary.jpg* from the *tutorial\tut02* directory in the *File Open* dialog that appears and click *Open*. SMS opens the file and searches for image georeferencing data. Georeferencing data define the world locations (x, y) that correspond to each point in an image. It is usually contained inside a world file or sometimes the image itself. A world file could have the extension “.wld”, “.tfw”, “.jpgw” ... If SMS finds georeferencing data, the image will be opened and displayed. If not, the user must define this mapping using the *Register Image* dialog. This is not required in this tutorial. Note that an entry is added to the *Project Explorer* as the image is read in under “Images”.
3. Depending on your preference settings, SMS may ask whether you want to build image pyramids. This improves image quality at various resolutions, but uses more memory. If asked, generate the pyramids.

2.5 Using Feature Objects

A conceptual model consists of a simplistic representation of the situation being modeled. This includes the geometric attributes of the situation (such as domain extents), the forces acting on the domain (such as inflow or water level boundary conditions) and the physical characteristics (such as roughness or friction). It does not include numerical details like elements. This model is constructed over a background image using *feature objects* in the Map  module.

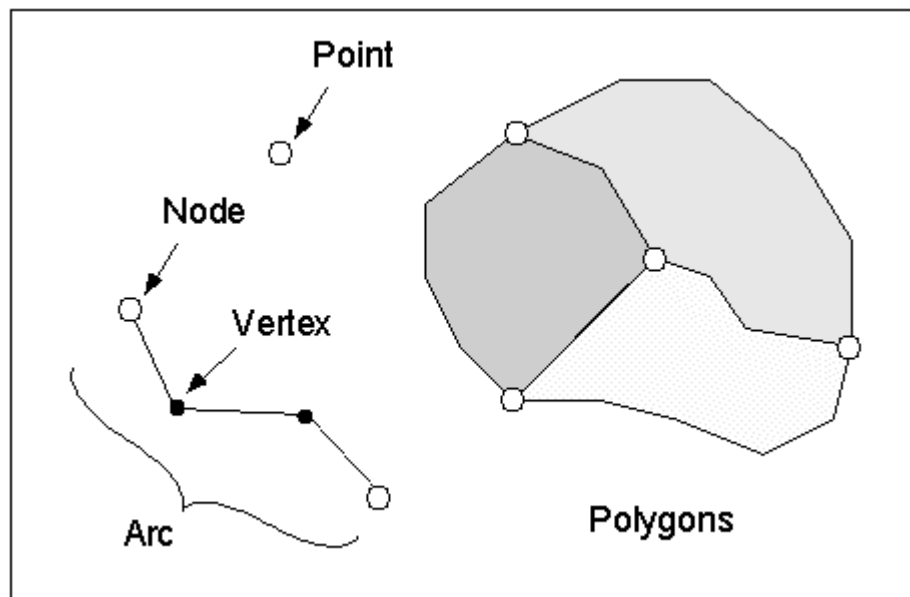


Figure 2-2 Feature Objects

Feature objects in *SMS* include points, nodes, arcs and polygons, as shown in Figure 2-2. Feature objects are grouped into sets called *coverages*. Only one coverage is active at a time.

A *feature point* defines an (x, y) location that is not attached to an arc. Points are used to define the location of a measured field value or a specific location of interest such as a velocity gauge. *SMS* can extract data from a numerical model at such a location, or force the creation of a mesh node at the specific location.

A *feature node* is the same as a feature point, except that it is attached to at least one arc.

A *feature arc* is a sequence of line segments grouped together as a polyline entity. Arcs can form polygons or represent linear features such as channel edges. The two end points of an arc are called *feature nodes* and the intermediate points are called *feature vertices*.

A *feature polygon* is defined by a closed loop of feature arcs. A feature polygon can consist of a single feature arc or multiple feature arcs, as long as a closed loop is formed. It may also include holes.

The conceptual model in this tutorial will consist of a single coverage, in which the river regions and the flood bank will be defined. As you go along in this tutorial you will load new coverages over the existing coverage. The new coverage will become active and the old coverage will become inactive.

2.6 Creating Feature Arcs

A set of feature objects can be created to show topographically important features such as river channels and material region boundaries. Feature objects can be digitized directly inside *SMS*, converted from an existing *CAD* file (such as *DXF* or *DWG*) or they can be extracted from survey data. For this example, the feature objects will be digitized inside *SMS* using the registered *JPEG* image as a reference. To create the feature arcs by digitizing:

1. Choose the *Create Feature Arc*  tool from the *Toolbox*.
2. Click out the left riverbank, as shown in Figure 2-3. As you create the arc, if you make a mistake and wish to back up, press the *BACKSPACE* key. If you wish to abort the arc and start over, press the *ESC* key. Double-click the last point to end the arc.

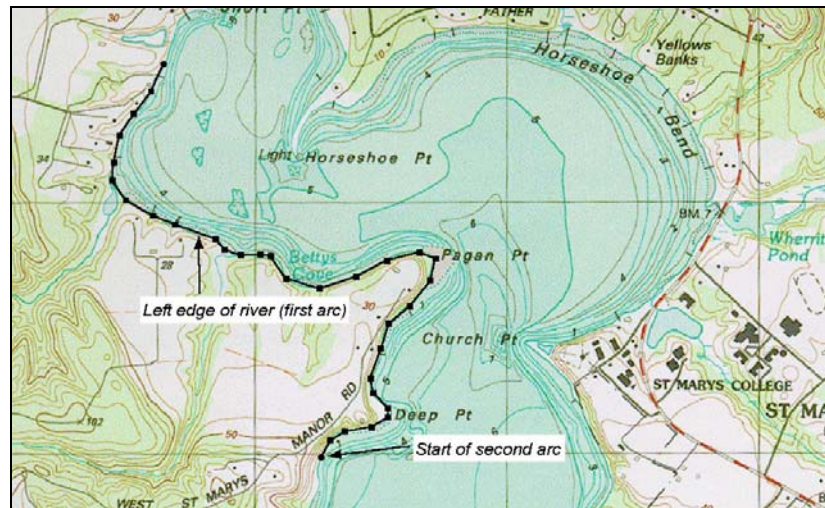


Figure 2-3 Creation of the first feature arc.

A feature arc has defined the general shape of the left riverbank. Three more arcs are required to define the right riverbank and the upstream and downstream river cross sections. Together, these arcs will be used to create a polygon that defines the study area. To create the remaining arcs:

- In the same manner just described, create the remaining three arcs, as shown in Figure 2-4. Remember to double-click to terminate an arc unless you are terminating at an existing node.

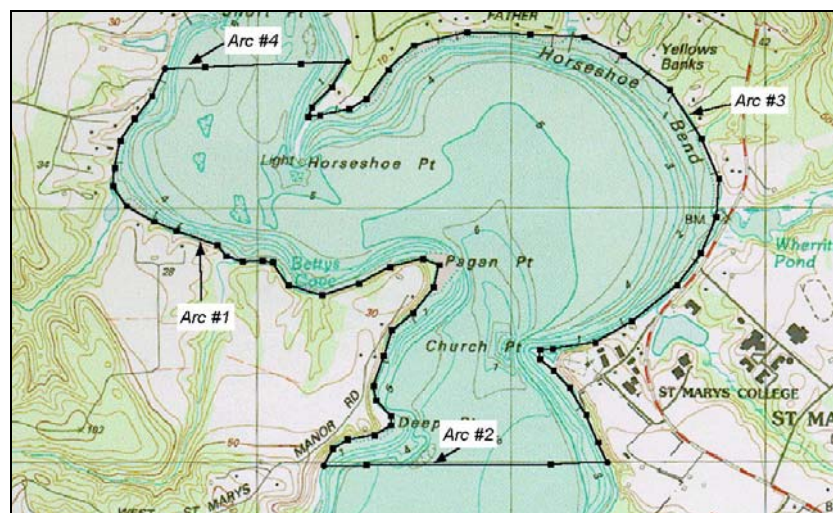


Figure 2-4 All feature arcs have been created.

You have now defined the main river channel. When creating your own models, you will proceed to create other arcs, and split the existing arcs to define material zones and locate specific model features such as hard points on the river. To save time, a conceptual model with this all done has been saved in a file. To open the file:

1. Select *File* | *Open*.

2. Open the file *stmary1.map* from the *tutorial\tut02* directory.

A new coverage is created from the data in the file, and the coverage you were editing becomes inactive. To hide the inactive coverage, uncheck the box next to its name (default coverage) in the *Project Explorer*. The new coverage is added to the *Project Explorer* with the name “stmary1”. The display should look something like Figure 2-5.

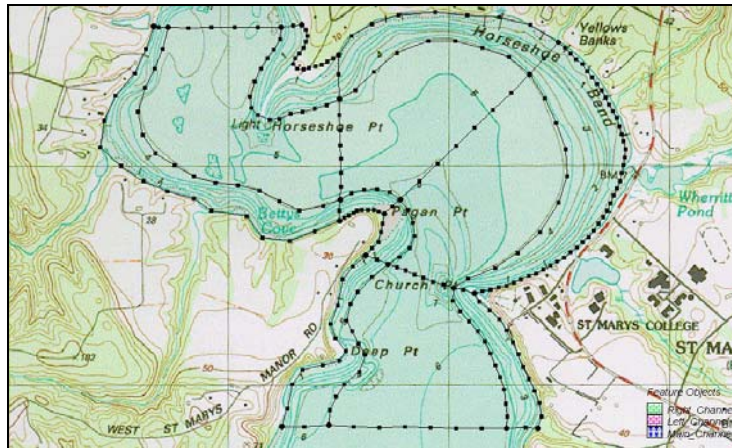


Figure 2-5 The *stmary1.map* feature object data.

2.7 Manipulating Coverages

As stated at the beginning of this tutorial, feature objects are grouped into coverages. When a set of feature objects is opened from a file, one or more new coverages are created. The last coverage in the file becomes active. Any creation or editing of feature objects occurs in the active coverage. Inactive coverages are drawn in a blue-gray color by default, or not displayed at all depending on the display attribute settings. Each coverage is also represented by an entry on the *Project Explorer*. A project commonly could include many coverages defining various options in a design or various historical conditions. When there are many coverages being drawn, the display can become cluttered. Individual coverages may be turned off by unchecking the box next to the coverage name in the *Project Explorer*. If a coverage is no longer desired, you may also delete it by right clicking on the coverage in the *Project Explorer* and selecting the *Delete* option.

2.8 Redistributing Vertices

To create the feature arcs, you simply clicked out a line of points on the image. You may or may not have paid much attention to the spacing of the vertices along the arc. The final element density in a mesh created from feature objects matches the density of vertices along the feature arcs, so it is desirable to have a more uniform node

distribution. The vertices in a feature arc can be redistributed at a desired spacing. To redistribute vertices:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Click on the arc to the far right, labeled *Arc #1* in Figure 2-6.
3. Select *Feature Objects | Redistribute Vertices*. The *Redistribute Vertices* dialog shows information about the feature arc segments and vertex spacing.
4. Choose the *Specified Spacing* option and enter a value of 470. This tells SMS to create vertices 470 ft apart from each other. If you are working in metric units, this would tell SMS to create vertices spaced 470 meters apart.
5. Click *OK* to redistribute the vertices along the arc.

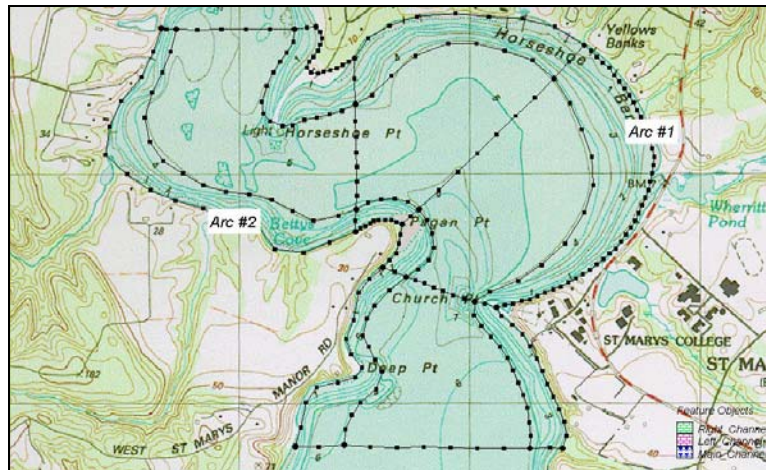


Figure 2-6 Redistribution of Vertices along arcs.

After clicking the *OK* button, the display will refresh, showing the specified vertex distribution. The arc will still be highlighted, because it is still selected. By clicking somewhere else on the display, the selection is cleared and the effect of the command can be more clearly seen.

When you create conceptual models, this redistribution would be done for each arc until you have the vertex spacing that you want in all areas. If the spacing is the same for multiple arcs, multiple arcs can be selected and redistributed at the same time. When you plan to use arcs in a patch, a better patch is created if opposite arcs have an equal number of vertices. In this case, you would want to use the *Number of Segments* option rather than the *Specified Spacing* option so that you can specify the exact number of vertices along each arc.

2.9 Defining Polygons

For this tutorial, you should open another map file, which has the vertices redistributed on all the arcs. Open the map file *stmary2.map* as you did the previous one and turn off the display of the “stmary1” coverage.

Polygons are created from a group of arcs that form a closed loop. Each polygon is used to define a specific material zone. Polygons can be created one by one, but it is more reliable to have SMS create them automatically. To have SMS build polygons out of the arcs:

1. Make sure no arcs are selected by clicking in the *Graphics Window* away from any arcs.
2. Select *Feature Objects | Clean* to be sure that there are no problems with the feature objects that were created. Click *OK* in the *Clean Options* dialog.
3. Select *Feature Objects | Build Polygons*.

Although nothing appears to have changed in the display, polygons have been built from the arcs. The one evidence of this is that the *Select Polygon* tool  becomes available (undims). The polygons in this example are for defining the material zones as well as to aid in creating a better quality mesh.

2.10 Assigning Meshing Parameters

With polygons, arcs and points created, meshing parameters can be assigned. These meshing parameters define which automatic mesh generation method will be used to create finite elements inside the polygon. For each method, a corner node of a finite element mesh will be created at each vertex on the feature arc. The difference comes in how internal nodes are created, and how those nodes are connected to form elements.

SMS has various mesh generation methods: patch, paving, scalar paving density, adaptive tessellation, and adaptive density. These methods are described in the *SMS online Help*, so they will not be described in detail here. As an overview, paving is the default technique because it works for all polygon shapes. Patches require either 3 or 4 polygonal sides. Density meshing options require scattered data sets to define the mesh density.

2.10.1 Creating a Refine Point for Paving

When using the default paving method, some control can be maintained over how elements are created. A *refine point* is a feature point that is created inside the boundary of a polygon and assigned a size value. When the finite element mesh is created, a corner node will be created at the location of the refine point and all

element edges that touch the node will be the exact length specified by the refine point size value. To create a refine point:

1. Choose the *Select Feature Point*  tool from the *Toolbox*.
2. Double-click on the point inside the left polygon, labeled in Figure 2-7.
3. In the *Feature Point/Node Attributes* dialog, make sure the *Refine Point* option is on and enter a value of 75.0 (ft).
4. Click the *OK* button to accept the refine point.

When the finite element mesh is generated, a mesh corner node will be created at the refine point's location, and all attached element edges will be 75.0 feet in length. A refine point is useful when a node needs to be placed at a specific feature, such as at a high or low elevation point.

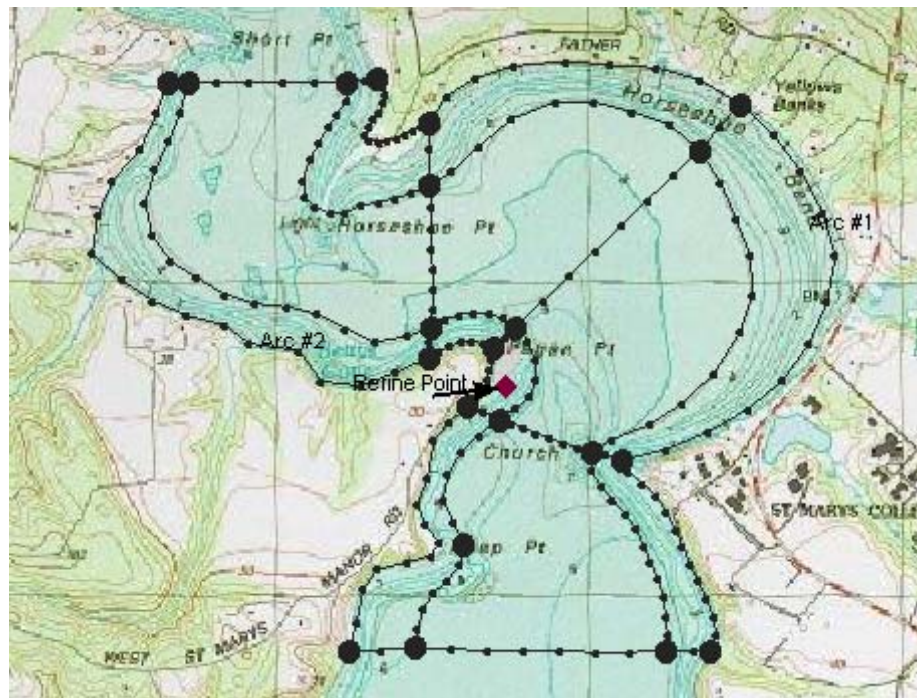


Figure 2-7 The location of the refine point.

2.10.2 Defining a Coons Patch

As was previously stated, the *Coons Patch* mesh generation method requires three or four sides to be created. However, it is not uncommon, that we wish to use the patching technique to fill a polygon defined by more than four arcs. Figure 2-8 shows an example of a rectangular patch made up of four sides. Note that *Side 1* and *Side 2* are both made from multiple feature arcs.

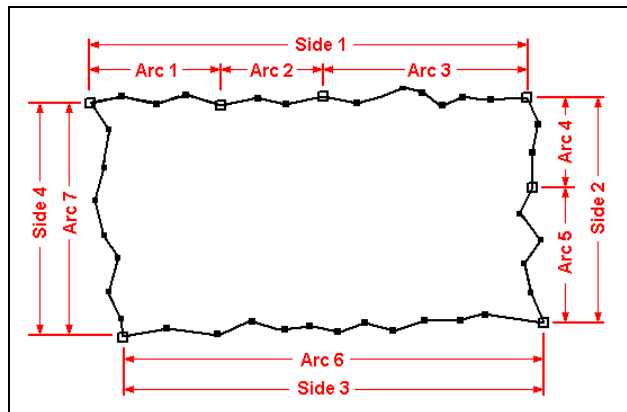


Figure 2-8 Four sides required for a rectangular patch.

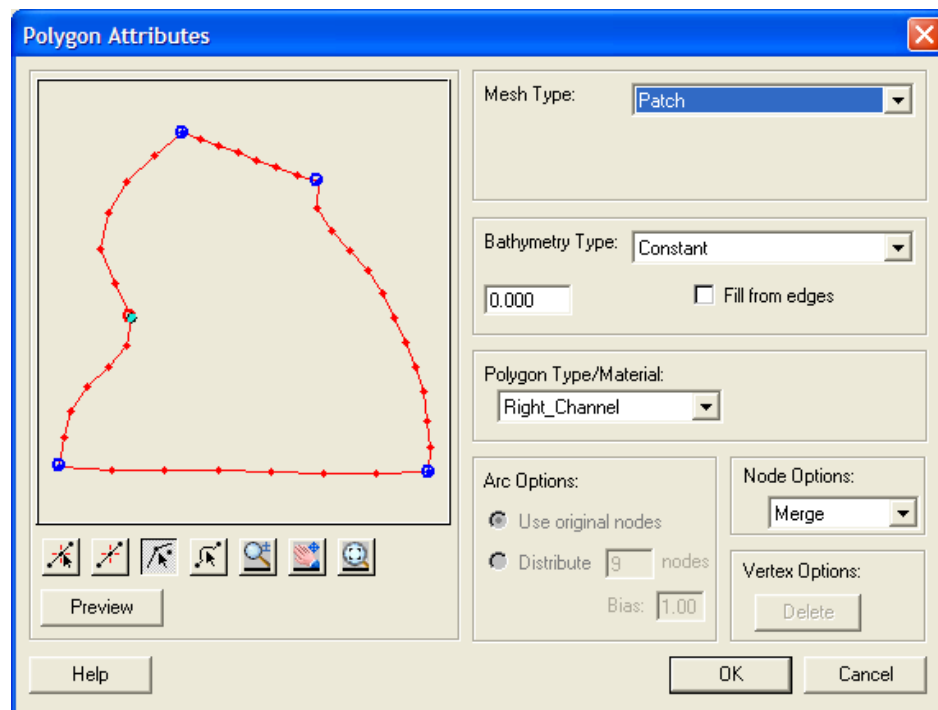


Figure 2-9 The Feature Polygon Attributes dialog.

SMS provides a way to define a patch from such a polygon by allowing multiple arcs to act as one. For example, the bottom middle polygon in our example, contains five arcs, but it should be used to create a patch. To do this:

1. Choose the *Select Feature Polygon*  tool and double click on the bottom middle polygon.
2. In the *Polygon Attributes* dialog, choose the *Select Feature Node*  tool.
3. Click on the node at the center of the left side, as seen in Figure 2-9.

4. Select the *Merge* option from the *Node Options* drop down list. This makes the two arcs on the left side be treated as a single arc.
5. Select the *Patch* option from the *Mesh Type* drop down list. (If you tried to assign the meshing type to be *Patch* before merging the node, *SMS* would have popped up a message box informing you that you need 3 or 4 sides for a patch.) If you wish to preview the patch, click the *Preview* button.
6. Click the *OK* button to close the *Polygon Attributes* dialog.

When you are creating your models, you will need to set up the desired polygon attributes for each feature polygon in your model. For this tutorial, the rest of the polygons have been set up for you and saved to a map file. To import this data:

- Open the file *stmary3.map*.

In the coverage that opens, all polygon attributes have been assigned. The three main channel polygons are assigned as patches, while the other polygons are assigned as adaptive tessellation.

2.10.3 Removing Drawing Objects

Throughout this tutorial, *drawing objects*, such as labels and arrows, have been provided to give a description of certain feature objects. Drawing objects are not part of a coverage, so they do not become inactive. The drawing objects that have thus far been used will not be needed anymore. To delete the drawing objects:

1. Choose the *Select Drawing Objects*  tool from the *Toolbox*.
2. Choose *Edit | Select All* to select all drawing objects.
3. Press the *DELETE* key or go to *Edit|Delete..*
4. If *Edit | Confirm Deletions* is on click *Yes*.

2.11 Applying Boundary Conditions

The coverage type controls which model will be used when a numeric model is generated from a conceptual model. This also controls the types of boundary conditions that can be assigned to the conceptual model. To view the type of the coverage, right click on the coverage in the *Project Explorer* and select type. The type associated with the selected coverage includes a check mark in the menu that appears. For this tutorial, make sure the type is selected as “Tabs” or “FESWMS”.

Boundary conditions can be assigned to arcs, points, and for *FESWMS*, polygons. Feature arcs may be assigned a flow, head or flux status. Feature points may be

assigned velocity or head values. Feature polygons may be assigned ceiling elevation functions, but only in a *FESWMS* coverage.

The inflow for this example is across the top of the model and the outflow is across the bottom. Notice that there are three feature arcs across each of these sections. A flow rate value could be assigned to each of the arcs at the inflow. However, this would create three separate inflow nodestrings, connected end-to-end. The same situation exists at the outflow cross section.

Both *RMA2* and *FESWMS* can have numerical problems if two boundary conditions are adjacent to each other with no corner between them. To avoid creating three separate boundary conditions at a single cross section, an *arc group* should be defined. An arc group consists of multiple arcs that are linked together. The arc group can be assigned the boundary condition instead of assigning it at the individual arcs so that when the model is generated, only a single nodestring is created, which spans the entire cross section.

2.11.1 Defining Arc Groups

For this example two arc groups will be defined. One will be positioned at the inflow boundary and one at the outflow boundary. To create the arc groups:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Holding the *SHIFT* key, select the three arcs that make up the flow cross-section, labeled as *Flow Arcs* in Figure 2-10. (Alternatively, you can select all three arcs by dragging a box around them. This box must include the entire arc.)
3. Select *Feature Objects* | *Create Arc Group* to create an arc group from the three selected arcs.
4. Now, select the three arcs that make up the head cross-section, labeled as *Head Arcs* in Figure 2-10. (Make sure only these three arcs are selected by checking the *Status Bar* at the bottom right of the *Main Graphics Window*.)
5. Select *Feature Objects* | *Create Arc Group*.

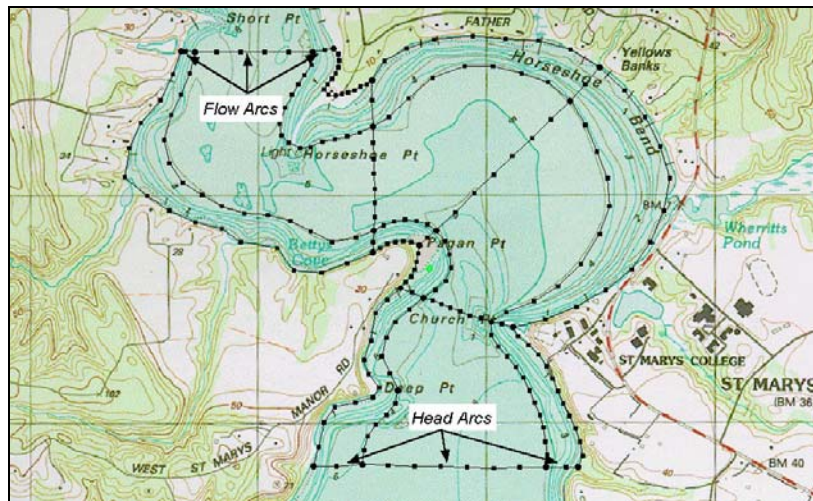


Figure 2-10 The arc groups to create.

2.11.2 Assigning the Boundary Conditions

With the arc groups created, boundary conditions can now be assigned. To assign the inflow boundary condition:

1. Choose the *Select Feature Arc Group*  tool from the *Toolbox*.
2. Double-click the arc group at the inflow (top) cross section.
3. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button.
4. If using *FESWMS*, the *FESWMS Nodestring Boundary Conditions* dialog will appear. Select the *Flow* option under *Specified Flow/WSE Options*. Enter in a *Constant* flowrate of 40,000 cfs.
5. If using *RMA2*, the *RMA2 Assign Boundary Conditions* dialog will appear. Select *Specified flowrate* as the *Boundary Condition Type*. Enter in a *Constant* flowrate of 40,000 cfs. Click the *Perpendicular to boundary* button to force the flow to enter the mesh perpendicular to the inflow boundary.
6. Click the *OK* button in both dialogs.

To assign the water surface boundary condition:

1. Double-click the arc group at the outflow cross section.
2. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button.

3. If using *FESWMS*, select the *Water surface elevation* option under *Specified Flow/WSE Options*. Enter in a *Constant* water surface elevation of 20 ft
4. If using *RMA2*, select *Water surface elevation* as the *Boundary Condition Type*. Enter in a *Constant* water surface elevation of 20 ft.
5. Click the OK button in both dialogs.

The inflow and outflow boundary conditions are now defined in the conceptual model. When the conceptual model is converted to a finite element mesh, SMS will create the nodestrings and assign the proper boundary conditions.

2.12 Assigning Materials to Polygons

Each polygon is assigned a material type. All elements generated inside the polygon are assigned the material type defined in the polygon. To assign the materials:

1. Choose the *Select Feature Polygon*  tool from the *Toolbox*.
2. Double-click on any of the polygons.
3. In the *Polygon Attributes* dialog, make sure the *Polygon Type/Material* shows the correct material for the polygon, as shown in Figure 2-11.
4. Click the *OK* button to close the *Polygon Attributes* dialog.

Repeat these steps to make sure the correct material type is assigned to each of the feature polygons. The following figure shows the materials that should be assigned to each polygon.

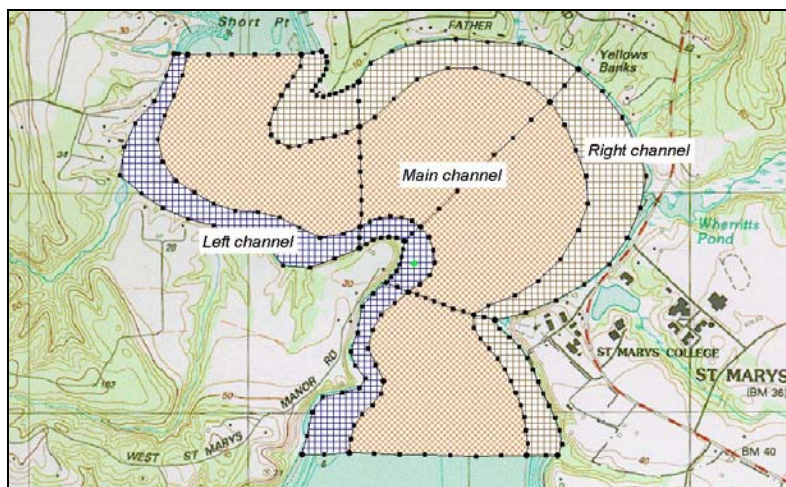


Figure 2-11 Polygons with defined material types.

2.12.1 Displaying Material Types

With the materials assigned to the polygons, you can fill the polygons with the material colors and patterns. To do this:

1. Click the *Display Options*  macro from the *Toolbox*.
2. If not active, select the *Map* tab in the *Display Options* dialog.
3. Turn on the *Polygon fill* option and select the *Fill with materials* option.
4. Click the OK button to close the *Display Options* dialog.

The display will refresh, filling each polygon with the material color and pattern.

2.13 Converting Feature Objects to a Mesh

With the meshing techniques chosen, boundary conditions assigned, and materials assigned, we are ready to generate the finite element mesh. To do this:

1. We want to convert the entire conceptual model to a mesh. Therefore, nothing should be selected. If individual polygons were selected, only those polygons would be converted to mesh segments. Make sure no objects are selected by clicking in the *Graphics Window* away from the river channel.
2. Select *Feature Objects* | *Map -> 2D Mesh*.
3. Click the *OK* button to start the meshing process.
4. Click the *No* button to not save a copy of the coverage.

After a few moments, the display will refresh to show the finite element mesh that was generated according to the preset conditions. With the mesh created it is often desirable to delete or hide the feature arcs and the image. To hide the feature arcs:

1. Click the *Display Options*  macro from the *Toolbox*.
2. If it is not active, select the *Map* tab.
3. Turn off the display of *Arcs*, *Nodes*, and *Polygon fill*.
4. Click the *OK* button to close the *Display Options* dialog.

To hide the image click on the toggle box next to the “stmary” image icon in the *Project Explorer*.

The display will refresh to show the finite element mesh, as shown in Figure 2-12. With the feature objects and image hidden, the mesh can be manipulated without interference, but they are still available if mesh reconstruction is desired.

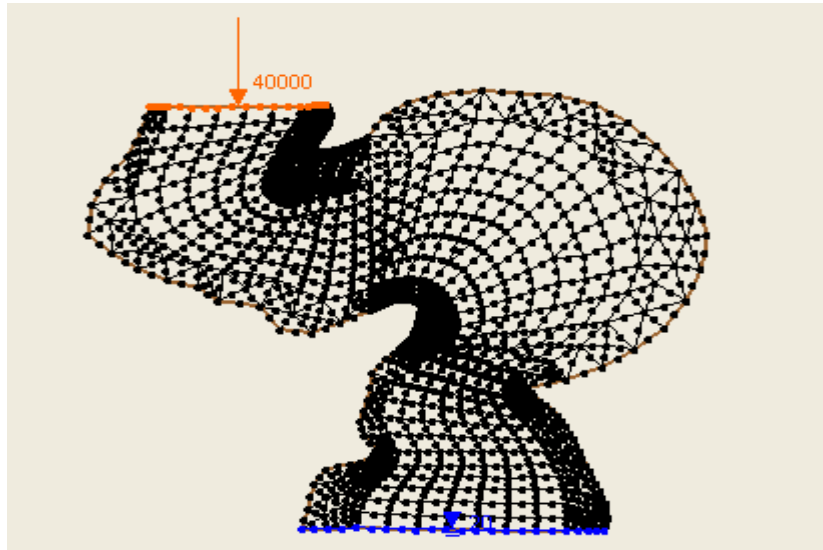


Figure 2-12 The finite element mesh that was created.

2.14 Editing the Generated Mesh

When a finite element mesh is generated from feature objects, it is not always the way you want it. An easy way to edit the mesh is to change the meshing parameters in the conceptual model, such as the distribution of vertices on feature arcs or the mesh generation parameters. Then, the mesh can be regenerated according to the new parameters. If there are only a few changes desired, they can be edited manually using tools in the mesh module. These tools are described in the *SMS Help* in the section on the *Mesh Module*.

2.15 Interpolating to the Mesh

The finite element mesh generated from the feature objects in this case only defined the (x, y) coordinates for the nodes. This is because we had not read in the bathymetric data before generating the mesh. Normally, you would read in the survey data, and associate it with the polygons to assign bathymetry to your model. However, to illustrate how to update bathymetry for an existing mesh, this section is included.

Bathymetric survey data, saved as scatter points can be interpolated onto the finite element mesh. To open the scattered data:

1. Select *File* | *Open* and open the file *stmary_bathy.h5*.

The screen will refresh, showing a set of scattered data points. Each point represents a survey measurement. Scatter points are used to interpolate bathymetric (or other) data onto a finite element mesh. Although this next step requires you to manually interpolate the scattered data, this interpolation can be set up to automatically take place during the meshing process. To interpolate the scattered data onto the mesh:

1. Make sure you are in the *Scatter*  module.
2. Select *Scatter* | *Interpolate to Mesh*.
3. In the *Interpolation* dialog, select *Linear* from the *Interpolation* drop down list. (For more information on SMS interpolation options, see the *SMS online Help*.)
4. Turn on the *Map Z* option at the lower left area of the dialog.
5. Click the *OK* button to perform the interpolation.

The scattered data is triangulated when it is read into SMS and an interpolated value is assigned to each node in the mesh. The *Map Z* option causes the newly interpolated value to be used as the nodal Z- coordinate.

As with the feature objects, the scattered data will no longer be needed and may be hidden or deleted. To hide the scatter point data uncheck the box next to the scatter set named “stmary_bathy” in the Project Explorer. To delete the scatter set, right click on this object and select *Delete*.

2.16 Renumbering the Mesh

The process of creating and editing a finite element mesh can cause the node and element ordering to become disorganized. Renumbering the mesh can restore a good mesh ordering. (The mesh is renumbered after the mesh generation, but the mesh is renumbered from an arbitrary nodestring, which does not always give the best renumbering). To renumber:

1. Switch to the *Mesh*  module.
2. Choose the *Select Nodestring* tool  from the *Toolbox*.
3. Select the flow nodestring at the top of the mesh by clicking inside the icon that is at the middle of the nodestring.
4. Select *Nodestrings* | *Renumber*.

2.17 Saving a Project File

Much data has been opened and changed, but nothing has been saved yet. The data can all be saved in a project file. When a project file is saved, several files are saved. Separate files are created for the map, scatter and mesh data. The project file is a text file that references the individual data files. To save all this data for use in a later session:

1. Select *File | Save New Project*.
2. Save the file as *stmaryout.sms*.
3. Click the *Save* button to save the files.

2.18 Conclusion

This concludes the Overview tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

Data Visualization

3.1 Introduction

It is useful to view the geospatial data utilized as input and generated as solutions in the process of numerical analysis. It is also helpful to extract data along a line (profile or transect), or at a point from this geospatial data. This *visualization* increases the applicability and usefulness of the modeling process. In this lesson, you will learn how to import, manipulate and view solution data. You will need the geometry file *tribflood.geo* and the solution file *tribflood.sol* created by a RMA2. These files are in the *tutorials/tut03* directory.

3.2 Data sets

A geospatial data set has one or more numeric values associated with each node in a mesh, cell in a grid, vertex in a scatter set, etc. Scalar data sets have one value per location. Two-dimensional vector data sets have two values for every location (an x-component and a y-component). Examples of scalar data sets include bathymetry, water surface elevation, velocity magnitude, Froude number, energy head, concentration, bed change, wave heights and many more. Examples of vector data sets include observed wind fields, flow velocities, shear stresses, and wave radiation stress gradients.

Steady state data sets represent a numerical solution where nothing changes with time. Dynamic data sets have data at specific times (time steps) to represent a numerical solution that changes with time.

3.3 Open the Geometry and Solution Files

SMS opens all supported input and solution files using the *File | Open* command.

1. Select *File | Open*.
2. Open the file *tribflood.geo* from the *tutorial\tut03* directory. A *.geo file is an *RMA2* mesh file.

With the geometry opened, the solution can be imported. To import the solution file:

1. Select *File | Open*.
2. Open the file *tribflood.sol* from the *tutorial\tut03* directory. This is a file generated by *RMA2* which includes data sets for water depths and velocities. *SMS* computes water surface elevations and velocity magnitudes from these data sets.

SMS displays the data sets as contours and vectors. To be consistent:

1. Open the *Display Options*  dialog.
2. Under the *2D Mesh* tab, check the *Contours* and *Vectors* options. Also, turn off the *Nodes* option.
3. Under the *Contour Options* tab, select *Color Fill* as the *Contour Method*.
4. Under the *Vectors* tab, select *Scale length to magnitude* as the option for *Shaft Length*. Change the scaling ratio to 4.0. Close the *Display Options* dialog.

3.4 Creating New Data sets with the Data Calculator

SMS has a powerful tool called the *Data Calculator* for computing new data sets by performing operations on scalar values and existing data sets. In this example, a data set will be created which contains the Froude number at each node. The Froude number is given by the equation:

$$\text{Froude Number} = \frac{\text{Velocity Magnitude}}{\sqrt{\text{gravity} * \text{water depth}}}$$

To create the Froude number data set:

1. Select *Data | Data Calculator*.
2. Highlight the *velocity mag* data set.

3. Under the *Time Steps* section, turn on the *Use all time steps* option and click the *Add to Expression* button. The *Expression* will show “d:all”. The letter ‘d’ corresponds to the *velocity mag* data set and ‘all’ signifies all time steps
4. Click the *divide* “ / “ button.
5. Click the *sqrt(x)* operation. The “??” text is just a placeholder to make sure you know that something should be placed there. It should be highlighted. Enter 32.2 for the constant *g*.
6. Click the *multiply* button “or type * in the equation”, then highlight the *water depth* data set and click the *Add to Expression* button.
7. The expression should now read: “d:all / sqrt(32.2 * e:all)”, where ‘d’ represents the velocity data set and ‘e’ represents the water depth data set. (This expression could also just be typed in directly.)
8. In the *Result* field, enter the name *Froude* and then click the *Compute* button. *SMS* will take a few moments to perform the computations. When it is done, the *Froude* data set will appear in the *Data Sets* window.
9. Click the *Done* button to exit the *Data Calculator* dialog.
10. Since the *Froude* data set is associated with the solution. Drag it into the *tribflood.sol* folder in the *Project Explorer*.

This data set can be contoured and edited with any of the other tools in *SMS*. It can be treated just as any other dynamic scalar data set and can be saved in a generic data set file. See the *SMS Help* for more information on saving data sets.

3.5 Contours

3.5.1 Turning on Contours

SMS provides several contour options to help visualize data sets. For this example, we will create contours for the velocity magnitude data set. To create contours for the velocity magnitude data set:

1. Switch to the velocity magnitude data set by choosing *velocity mag* in the *Project Explorer*. Set the time step to 0 in the *Time Step* list-box below the *Project Explorer*.
2. Click on the *Display Options* macro .
3. Click *All off* to turn off all existing display options.

4. Turn on the *Mesh boundary*, the *Wet/dry boundary* and *Contours*.
5. Select the *Contour Options* tab.
6. Set the *Contour Method* to *Normal Linear* and the *Number of Contours* to 20.
7. Click *OK*.

In addition to linear contours, *SMS* also supports color-filled contours as well as color-filled with linear contours at the breaks. To turn use color-filled contours:

1. Right click on the “Mesh Data” item in the *Project Explorer* and select the *Display Options* command (this is an alternative to using the macro).
2. Switch to the *Contour Options* tab.
3. Change the *Contour Method* to *Color Fill*.
4. Make sure the *Fill continuous color range* option located at the bottom right side of the dialog is on. This toggle causes *SMS* to blend data set values rather than use discreet intervals.
5. Click *OK* to see color-filled contours on the mesh.

3.5.2 Color Ramp Options

The default color ramp in *SMS* has dark blue for the largest scalar value to a dark red for the smallest scalar value. Other color ramps can be useful for visualizing data and can be saved as part of a project or as the user’s default when running *SMS*. To use a different color-ramp to better visualize water depths:

1. Switch to the water depth data set by choosing *water depth* in the *Project Explorer*.
2. Bring up the *Display Options* (by either method already used).
3. Select the *Contour Options* tab.
4. Click on the *Color Ramp* button.
5. Select the *User defined* radio button.
6. Click the *New Palette* button.
7. Change the *Initial Color Ramp* to *Ocean*.
8. Click *OK* three times to get back to the main *SMS* screen.

This color ramp shows the deeper areas as dark blue and shallower areas as light blue.

3.6 Vectors

Vector data sets can be visualized inside of *SMS* by displaying arrows representing the direction and optionally the magnitude of vector data set over the mesh.

To turn on vectors for the velocity data set:

1. Switch to the velocity magnitude data set by choosing *velocity mag* in the *Project Explorer*.
2. Bring up the *Display Options*.
3. Click the toggle labeled *Vectors* on the *2D Mesh* tab.
4. Switch to the *Contour Options* tab.
5. Click on the *Color Ramp* button and change back to a *Hue ramp*. Click on the *Reverse* button at the bottom of the dialog to make red indicate the higher velocities. Click *OK*.
6. Select the *Vectors* tab.
7. In the *Shaft Length* section choose *Define min and max length*. This scales the length of the arrows based upon the magnitude of the velocity data set at the arrow location. The minimum data set magnitude uses the shaft length that is the minimum length. Likewise the maximum data set magnitude uses the maximum shaft length. Enter values of 10 and 80 in the two fields.
8. Click *OK*.

Arrows should now be displayed that show the magnitude and direction of the water currents over the mesh. However, the arrows are so dense that it is a black mess. To thin out the arrows:

1. Bring up the *Display Options* and click on the *Vectors* tab.
2. In the *Arrow Placement* section, choose “on a grid” and enter 25 in both of the “pix” edit fields. Enter a *Z-offset* of 5.0 and click *OK*.

Now the arrows should be evenly distributed over the domain at 25 pixel increments. The z-offset lifts the vectors off the mesh by 5.0 feet. Variations in the shape of the river bed can hide vectors since they are drawn in three dimensions.

Right below where the two branches join, an eddy is formed. Zoom in around the eddy as shown in Figure 3-1.

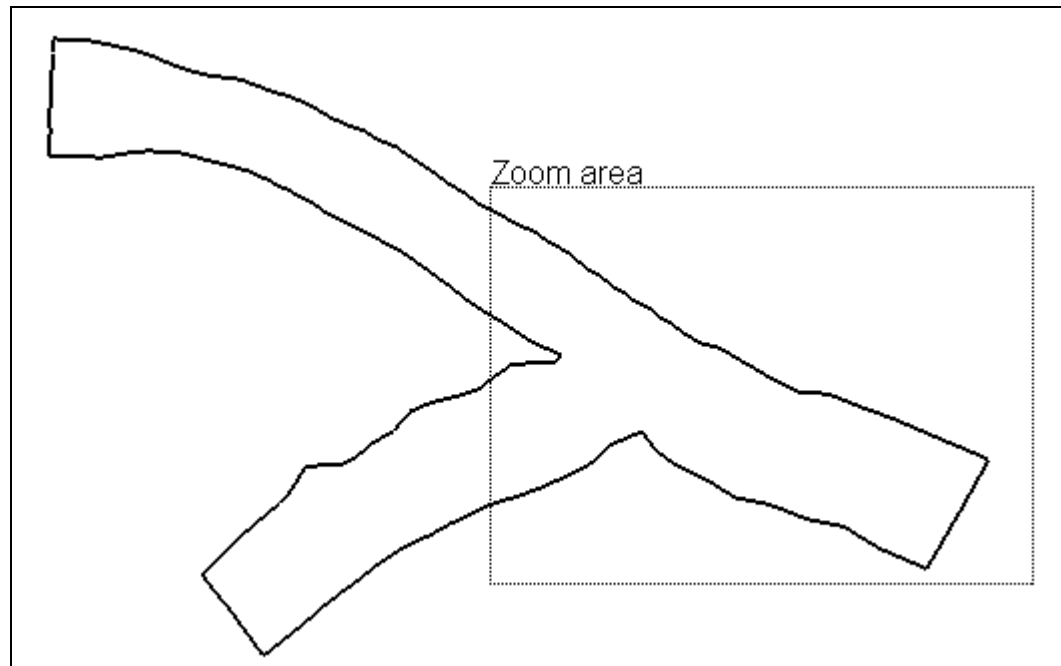


Figure 3-1 Area to zoom to.

As you zoom, the vector spacing stays at 25 pixels. Therefore, additional vectors appear illustrating the recirculation pattern.

3.7 Creating Animations ---

A film loop is an animation created by SMS to display changes in data sets through time. Flow trace and particle trace animations are a special type of film loop, which use vector data sets to trace the path that particles of water will follow through the flow system. Only the visible portion of the mesh will be included in the film loop when it is created

3.7.1 Creating a Film Loop Animation ---

The following film loop will show how the velocity changes through time. To create and run the film loop:

1. Make sure that the *velocity mag* (scalar) and *velocity* (vector) data sets are active by selecting them in the *Project Explorer*.
2. Select *Data | Film Loop*.
3. Make sure *Scalar/Vector Animation* is selected for the *Film Loop Type*.

4. Click the *File Browser*  button and enter the name “velocity.avi” and click *Save*. Then click *Next*.
5. In the *Time Step Options* page, turn on both the *Scalar Data Set* and *Vector Data Set* options. The other controls on this page allow the selection of part of the simulation, or the interpolation to create a smoother animation. All time steps should be selected, so click the *Next* button.
6. The last page of the setup allows the display options to be modified and a clock to be specified. Click on the *Clock Options* and change the “Location” to the “Top Right Corner”. Click the *OK* button and then click the *Finish* button in the *Film Loop Setup* wizard to create the film loop.

SMS will display each frame of the film loop as it is being created. When the film loop has been fully generated, it will launch in a new *Play AVI Application* (PAVIA) window. This application contains the following controls:

Play  button. This starts the playback animation. During the animation, the speed and play mode can be changed.

Speed. This increases or decreases the playback speed. The speed depends on your computer.

Frame. This control can be used to jump to a specific frame of an animation.

Stop  button. This stops the playback animation.

Step  button. This allows you to manually step to the next frame. It only works when the animation is stopped.

Loop  play mode. This play mode restarts the animation when the end of the film loop has been reached.

Back/forth  play mode. This play mode shows the film loop in reverse order when the end of the film loop has been reached.

The generated film loop illustrating a storm hydrograph coming down the tributary will be saved in the AVI file format. AVI files can be used in software presentation packages, such as Microsoft PowerPoint or WordPerfect Presentations. A saved film loop may be opened from inside SMS or directly from inside the PAVIA application. (pavia.exe is located in the SMS installation directory and can be freely distributed.)

3.7.2 Animating a Functional Surface

Functional surfaces can be used to visualize data sets as a surface with the elevation at each node being the value of the data set plus a constant offset. A functional surface can be used to display the water surface. To turn on the functional surface:

1. Bring up the *Display Options* dialog.
2. Turn off the vectors option.
3. Click on the *Functional Surface* toggle.
4. Click on the *Options* button to the right of the *Functional Surface* option.
5. In the *Data Set* section, select “User defined data set”. The *Select Data Set* dialog should open. Choose the “water surface elevation” data set. Then click *Select*.
6. Click the *Choose Color* button and change the functional surface color to a dark blue.
7. Click *OK* in the *Color Options* dialog and *OK* again in the *Functional Surface Options* dialog.
8. Switch to the *General* tab.
9. Change the *Z magnification* under drawing options to 5.0.
10. Click *OK*.
11. Select the *Display|View|View Angle* command to change the view to an oblique (3D) view. Enter a *Bearing* of 43, and *Dip* of 22, a *Look At Point* of (17275, 13900, 575) and a *Width* of 3200. Then click *OK*. (These values are selected to illustrate the oblique view, you can select any view you want using the rotate tool .)

The functional surface of water surface should appear over the bathymetry, shaded with the velocity magnitude contours. This surface can be animated to show the change in water surface elevation through time. In the case of this water surface, there are not huge changes. However, the water level does rise in the tributary and some flooding does occur. To view the animation:

- Follow the steps from the previous section to animate the functional surface.

3.7.3 Creating a Flow Trace Animation

A flow trace animation can be created if a vector data set has been opened. The flow trace simulates spraying the domain with colored dye droplets and watching the color

flow through the domain. Steady state vector fields can be used in a flow trace animation to show flow direction trends. For dynamic vector fields, the flow trace animation can trace a single time step, or it can trace the changing flow field.

Note that a flow trace generally takes longer to generate than the scalar/vector animation. As the window gets bigger and shows more of the model, the animation gets larger and requires more memory to generate it. If you have problems with this operation, decrease the size of the *SMS* window and try again. To create and run a flow trace film loop:

1. Click on the *Plan View*  button and then the *Frame*  button.
2. Zoom in on the area from the junction of the two reaches.
3. Select *Data | Film Loop*.
4. In the *Select Film Loop Type* section, select the *Flow Trace* option and change the file name to *flowtrace.avi*. Then click the *Next* button.
5. Make sure all the time steps of the velocity vector data set are selected and click the *Next* button.
6. Enter 0.5 as the number of “Particles per object” and 0.1 as the “Decay ratio”. Leave all other options in the *Flow Trace Options* page as their default values and click the *Next* button.
7. Click the *Finish* button to generate the animation.

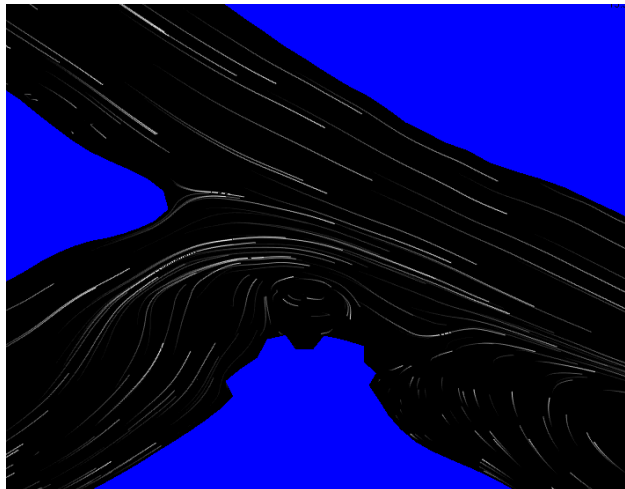


Figure 3-2. One frame from the *Id1* flow trace.

After a few moments, the first frame of the flow trace animation will appear on the screen. As before, the frames are generated one at a time, and a prompt shows which frame is being created. When the flow trace has been created, it is launched in a new window, just as the previous animation.

- View the flow trace using the same controls as with the film loop animation.

3.7.4 Droque Plot Animation

Droque plot animations are similar to flow trace animations, except that they allow the user to specify where particles will start. A particle/droque coverage defines the starting location for each particle. To create this coverage:

1. Click on the “default coverage” item in the *Project Explorer* to switch to the Map module. Then right click on the “default coverage” item and select the “Type” command. Change the type to *Particle/Droque*.
2. Create two feature arcs with the *Create Feature Arcs*  tool, one across each upstream branch of the river.
3. Select both arcs with the *Select Feature Arcs*  tool by holding *SHIFT* while clicking on each, and choose *Feature Objects | Redistribute Vertices*.
4. Change the *Specify* option to *Number of Segments* and set *Num Seg* to 20.
5. Click *OK* to close the *Redistribute Vertices* dialog.
6. Create three individual points in the downstream branch of the river with the *Create Points*  tool.

The arcs and points that were defined should look something like Figure 3-3. For the droque plot animation, one particle will be created at each feature point each feature arc vertex.

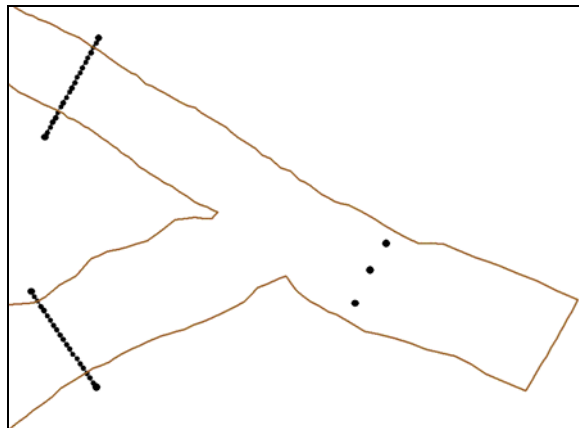


Figure 3-3. Feature objects for the particle/droque coverage.

To create the droque plot animation:

1. Click on the “Mesh” item to switch back to the *Mesh*  module and select *Data | Film Loop*.
2. Select the *Droque Plot* animation type and change  the filename to “drogue.avi”. Then click the *Next* button (the coverage was just created so it is already set).
3. In the Time Step section, set the Start Time to 0.0 and the End Time to 5.0. Set the simulation to generate 50 frames and click the *Next* button.
4. In the *Color Options* section, associate the color ramp with the *Distance traveled*, and set the *Maximum distance* to 1500. Turn on the *Write report* option, then click *Next*.
5. Click the *Finish* button to generate the animation.

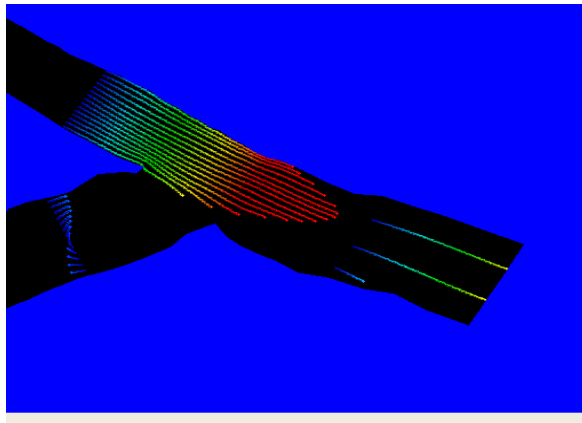


Figure 3-4. Sample drogue plot animation.

The drogue plot animation generates a report by turning on the option in step number 4 above. To see this report:

- Choose *File | View Data File* and Open the file “drogues.pdr”.

Now that you have seen the three main animation types available in SMS 9.0, feel free to experiment with some of the available options, especially with the flow trace and drogue plot animation types.

3.8 2D Plots

Plots can be created to help visualize the data. Plots are created using the observation coverage in the map module. Lesson 4 teaches how to use the observation coverage.

3.9 Conclusion

This concludes the tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to save, click the *No* button.

Observation Coverage

4.1 Introduction

An important part of any computer model is the verification of results. Surface water modeling is no exception. Before using a surface water model to predict results, the model must successfully simulate observed behavior. Calibration is the process of altering model input parameters (within an accepted range) until the computed solution matches observed field values (or at least as well as possible). *SMS* contains a suite of tools in the *Observation Coverage* to assist in the model verification and calibration processes.

The observation coverage consists of *Observation Points* and *Observation Arcs*, which help analyze the solution for a model. Observation points can be used to verify the numerical analysis with measured field data and calibration. They can also be used to see how data changes through time. Observation arcs can be used to view the results for cross sections or river profiles. This tutorial is based on a *FESWMS* finite element model, but the calibration tools in *SMS* can be used with any model.

4.2 Opening the Data

To open the *FESWMS* simulation and solution data:

1. Select *File* | *Open...*

2. Open the file *observe1.sms* from the *tut04* directory. If geometry data is still open from a previous tutorial, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.

4.3 Viewing Solution Data

An initial solution has already been created with this data file and was opened with the project. When the solution file is opened into *SMS*, various scalar and vector data sets are created. By default, the active data sets are the *velocity mag* scalar data set and the *velocity* vector data set. Several display options should be changed. To do this:

1. Right click on the *Mesh Data* object in the *Project Explorer* and select *Display Options*.
2. Click the *All off* button and then turn on the *Contours*, *Nodestrings* and *Mesh boundary* options.
3. Click *OK* to exit the *Display Options* dialog.

After setting the display options, the mesh data will appear as shown in Figure 4-1.

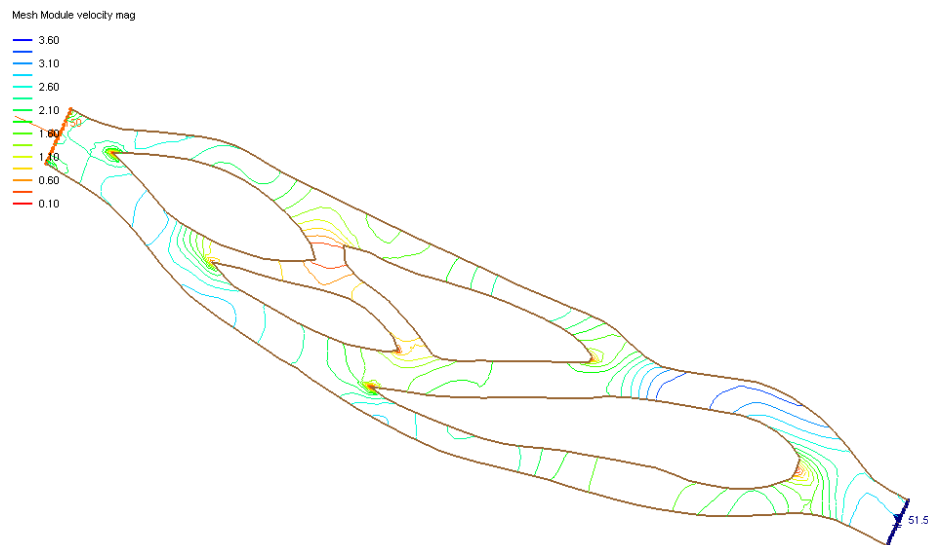


Figure 4-1 The mesh contained in *observe1.sms*

4.4 Creating an Observation Coverage

The calibration tools utilize observation features in an observation coverage. To create an observation coverage:

1. Click on the “default coverage” in the *Project Explorer* to make it the active object.
2. Right click on the “default coverage” and select *Rename*. Change the name to “calibration data”.
3. Right click on the coverage again and select *Type*. Change the type to “Observation”.

The *Observation Coverage* dialog can now be used to specify what data to use in calibrating the model and to edit observation points and arcs. To bring up the *Observation Coverage* dialog select *Feature Objects | Attributes...*

4.5 The Observation Coverage

In this tutorial, observation points will be used to calibrate the model; however, observation arcs or a combination of arcs and points can be used instead depending on the data collected in the field. Observation arcs work similar to observation points. Differences will be pointed out as the tutorial proceeds.

The *Observation Coverage* dialog can show the attributes for either observation points or observation arcs, but not both at the same time. The *Feature Object* combo box (in the upper right corner) determines which attributes are currently being shown in the *Observation Coverage* dialog.

The upper spreadsheet is called the *Measurements* spreadsheet and the lower spreadsheet is called the *Observation Objects* spreadsheet. The titles of these spreadsheets change depending on what is selected as the feature object. Right now, the title of the *Measurements* spreadsheet is simply “Measurements” and the title of the *Observation Objects* spreadsheet is “Observation Points.” Select *arcs* as the feature object and the titles of the *Measurements* and *Observation Objects* spreadsheets will change to “Flux Measurements” and “Observation Arcs,” respectively.

Before continuing, it should be pointed out that observation points use single values measured in the field such as velocity and water surface elevation to calibrate the model. On the other hand, observation arcs use fluxes that have been computed across the arc to calibrate the model. Therefore, measurements for observation arcs are called “Flux Measurements.”

4.5.1 Creating a Measurement

By default, when the *Observation Coverage* dialog is first opened, a *Measurement* does not exist. A measurement represents the solution data that is compared to the observed field data in the calibration process. For observation points, a measurement is tied to either a scalar or a vector data set. This data set is unique to the

measurement and cannot be tied to another measurement. For observation arcs, a measurement is tied to both a scalar and a vector data set. Again, this combination of data sets is unique to the measurement.

In addition to a unique *Name* and *Data Set(s)*, two other parameters are used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, select the *Trans* toggle. The *Module* of a measurement refers to the SMS module where the computed data is stored. (The *Module* is set by default and normally does not need to be changed.)

To create a new measurement:

1. Make sure *points* is selected as the feature object.
2. Type “Velocity” as the *Name* of the measurement.
3. Select *velocity* as the *Data Set* (not *velocity mag*).

Now that a measurement has been defined, observation points can be created and edited.

4.6 Creating an Observation Point

Observation points are created at locations in the model where the velocity or water surface elevation has been measured in the field. The measured values will be compared with the values computed by the model to determine the model’s accuracy. In addition to being assigned a *Color* and a *Name*, each observation point is assigned the following data:

Location. The x, y real world location of the point needs to be specified. Observation arcs do not have these location attributes since several points define an arc.

Observed value. The observed value is the value that was measured in the field corresponding to the active measurement.

Confidence Interval. The confidence interval is the allowable error ($\pm$) between the computed value and the observed value. Model verification is achieved when the error is within the interval ($\pm$) of the observed value.

Confidence Level. The percent of confidence that the mean of the observed value will lie within confidence interval.

Angle. When a measurement for observation points is tied to a vector data set (as is the case with the *Velocity* measurement created in the previous section) an angle needs to be specified. This angle is an azimuth angle with the top of the screen representing north when in plan view.

Table 4-1 Observation point values

x [ft]	y [ft]	Velocity [fps]	Interval [fps]	Confidence [%]
190	-369	3.5	0.25	95

One observation point should be created using the values in Table 4-1. In this case, the model will be verified if the computed value is ± 0.25 fps of the observed velocity, or between 3.25 and 3.75 fps. To create the observation point while still in the *Observation Coverage* dialog:

1. Type “Point 1” as the *Name* in the bottom line of the *Observation Points* spreadsheet. The *Observation Points* spreadsheet will always end with a blank line for the creation of additional points. (Note, there will be no blank line in the *Observation Arcs* spreadsheet since arcs cannot be created while in the *Observation Coverage* dialog.)
2. Press *Enter* or *Tab* to create the new observation point.
3. Now that the observation point had been created, change the *X* coordinate to 190.0 and the *Y* coordinate to -369.0.
4. Enter 3.5 as the *Observed Val* and 0.25 as the *Conf. Int*. The *Conf. (%)* is already be set to 95. By default, after the *Observed Val* is entered, the *Observe* toggle for this point turns on. When the *Observe* toggle for a point or arc is on, it is said to be *Observed*.

An observation point has now been created at the location specified in the *Observation Coverage* dialog. However, no angle has been specified for this point. This angle can be specified in the *Observation Coverage* dialog or in the *Graphics Window*. To specify the angle in the *Graphics Window*:

1. Click *OK* to close the *Observation Coverage* dialog. A point with an arrow pointing up will appear in the *Graphics Window*. A calibration target is drawn next to the point.
2. Choose the *Select Feature Point*  tool from the *Toolbox*.
3. Zoom in and rotate the point arrow approximately 120° by dragging the black dot at the end of the arrow clockwise. Do not worry if this angle is not exactly 120°. The arrow just needs to be pointing in the general direction the velocity meter was set up in the field. This is usually in the direction of flow. Figure 4-2 shows a close-up of *Point 1* with the arrow pointing up (0° angle) and then the position of the arrow at an angle of approximately 120°.

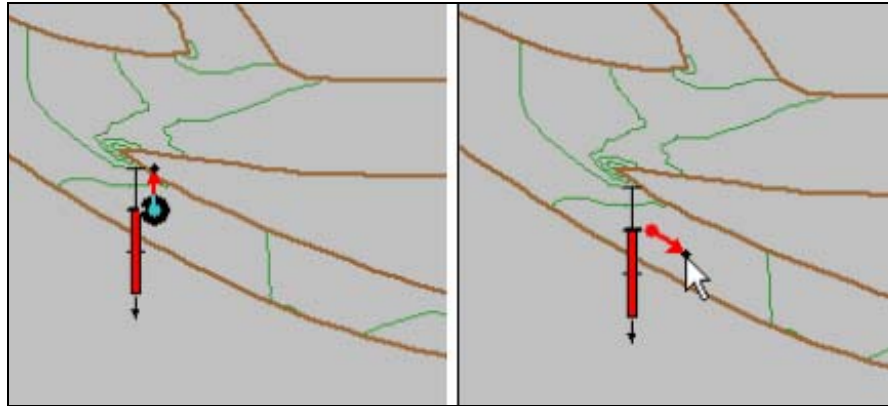


Figure 4-2 "Point 1" with an arrow angle of 0° and then rotated to 120°

4.6.1 Using the Calibration Target

A calibration target is drawn next to the observation point. The components of a calibration target are illustrated in Figure 4-3. These components are:

Target Middle. This is the target value that was measured in the field.

Target Extents. The top of the target represents the target value plus the interval while the bottom represents the target value minus the interval.

Color Bar. The color bar shows the error between the observed value and the computed value. If the bar is entirely within the target, the color bar is drawn in green. If the error is less than twice the interval, the bar is drawn in yellow. A larger error will be drawn in red.

For this example, the bar would be green if the computed value is between 3.25 and 3.75, yellow for values between 3.0-3.25 or 3.75-4.0, and red for values smaller than 3.0 or greater than 4.0.

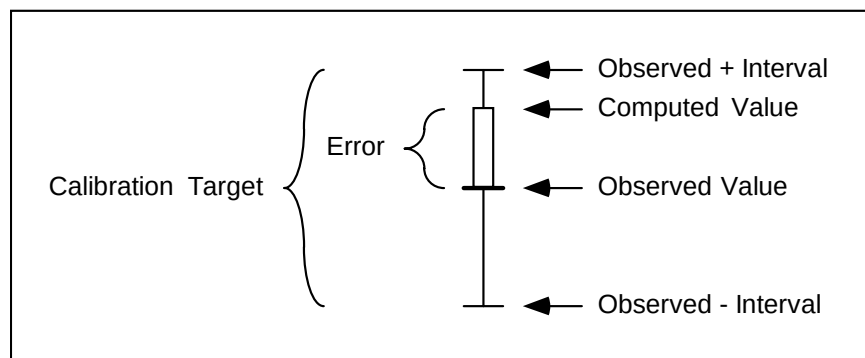


Figure 4-3 Calibration target

Now that the observation point has been created and a solution has been opened, the target appears. The color bar in this example is red with an arrow pointing down, indicating that the computed solution has a velocity below 3.0.

4.6.2 Multiple Measurements

Each observation point has attributes for all measurements. Similarly, each observation arc has attributes for each flux measurement. The highlighted measurement in the *Measurements* spreadsheet determines which attributes are shown in the *Observation Objects* spreadsheet.

For example, to create a new measurement:

1. Open the *Observation Coverage* dialog by choosing the *Select Feature Point*  tool from the *Toolbox* and double-clicking *Point 1*.
2. Type “WSE” as the *Name* in the bottom line of the *Measurements* spreadsheet. As with the *Observation Points* spreadsheet, the *Measurements* spreadsheet will always end with a blank line for the creation of additional measurements.
3. Press *Enter* or *Tab* to create the new measurement when finished typing to create the new measurement.
4. Select *water surface* as the *Data Set*.

Note that this new measurement is now the *Active* measurement and it is also highlighted. Several measurements can exist at a time; however, calibration targets will only be displayed in the *Graphics Window* for *Observed* points in the *Active* measurement.

Now look at the *Observation Points* spreadsheet. The *Name*, *Color*, and *X* and *Y* coordinates have remained the same for *Point 1*, however, the *Observed Val* and *Conf. Int.* have been reset to their default values. There is no *Angle* column as well since this new measurement is tied to a scalar data set. These attributes are for the measurement named *WSE*. To view the observation point attributes previously specified for the *Velocity* measurement, simply click the *Velocity* measurement to highlight it in the *Measurements* spreadsheet.

Do not delete the *WSE* measurement since both it and the *Velocity* measurement will be used to calibrate the model. Before continuing, make the *Velocity* measurement the *Active* measurement.

4.7 Reading a Set of Observation Points

Using the steps defined above, multiple observation points can be created. However, this process could become tedious for a large set of points. Normally, the data defining the points will be in spreadsheet format and can simply be copied and pasted in the *Observation Points* spreadsheet. To do this:

1. Open the file *observepts.xls* in a spreadsheet program. (This is a Microsoft Office/Excel file. If you prefer another spreadsheet, the data is also contained in a tab delimited file named *observepts.txt*.)
2. Highlight and copy the data from the column labeled “Name” to the first column labeled “int” for *Point 2* to *Point 8*. The data for *Point 1* does not need to be copied since *Point 1* has already been created.
3. Return to *SMS* and make sure the *Velocity* measurement is selected.
4. Select the *Name* of the bottom row of the *Observation Points* spreadsheet as the starting cell for the data to be pasted and paste the copied data into the *Observation Points* spreadsheet.
5. Click *OK* to close the *Observation Coverage* dialog.

Seven new observation points appear in the *Graphics Window*. The new points are distributed around the finite element mesh, as shown in Figure 4-4.

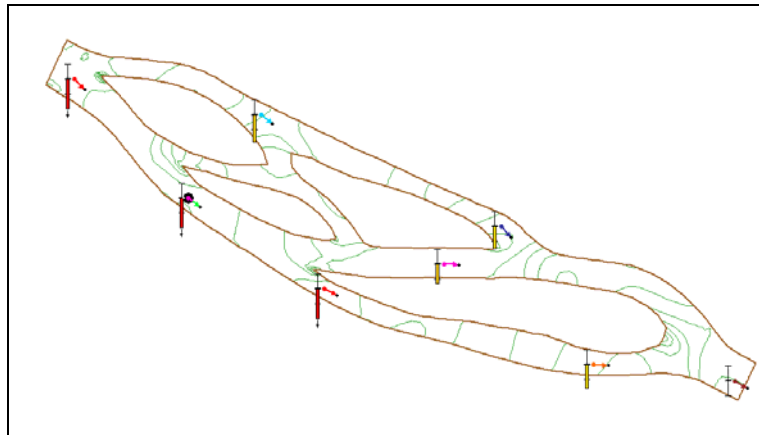


Figure 4-4 The observation points created from the file *observepts.obt*

Now, the observed values and the confidence interval for the *WSE* measurement need to be specified. To do this:

1. Open the *Observation Coverage* dialog by double-clicking on one of the points.

2. Using the same spreadsheet file opened earlier, *observepts.obt*, highlight and copy the data from the column labeled “wse” to the second column labeled “int” for *Point 1* to *Point 8*.
3. Return to *SMS* and make sure the *WSE* measurement is selected.
4. Select the *Observed Val* of the top row of the *Observation Points* spreadsheet as the starting cell for the data to be pasted and paste the copied data into the *Observation Points* spreadsheet.

To view that calibration targets for the *WSE* measurement, make the *WSE* measurement the *Active* measurement and close the *Observation Coverage* dialog by clicking the *OK* button. The points that appear in the *Graphics Window* do not have arrows since the active measurement is observing a scalar data set.

When calibrating a model the goal is to calibrate the model so that the computed values from the model fall within the confidence intervals of the observed field data for all measurements. At times this is difficult and personal discretion is required to determine when the model has sufficiently been calibrated. Before continuing, make the *Velocity* measurement the *Active* measurement.

4.8 Generating Error Plots

SMS can create several types of plots to analyze the error between the computed and observed values. To create a *Computed vs. Observed Data* plot and an *Error Summary* plot

1. Select *Display | Plot Wizard...* .
2. Choose *Computed vs. Observed Data* as the *Plot Type*.
3. Click *Next* and choose *Velocity* as the measurement.
4. Click *Finish* to close the *Plot Wizard* and generate the plot.

Create another plot of the *Velocity* measurement, but this time choose *Error Summary* as the *Plot Type*. Again choose *Velocity* as the measurement.

Both plots have now been created. Each plot exists in a separate window that can be resized, moved, and closed at any time. The plots that appear are shown in Figure 4-5.

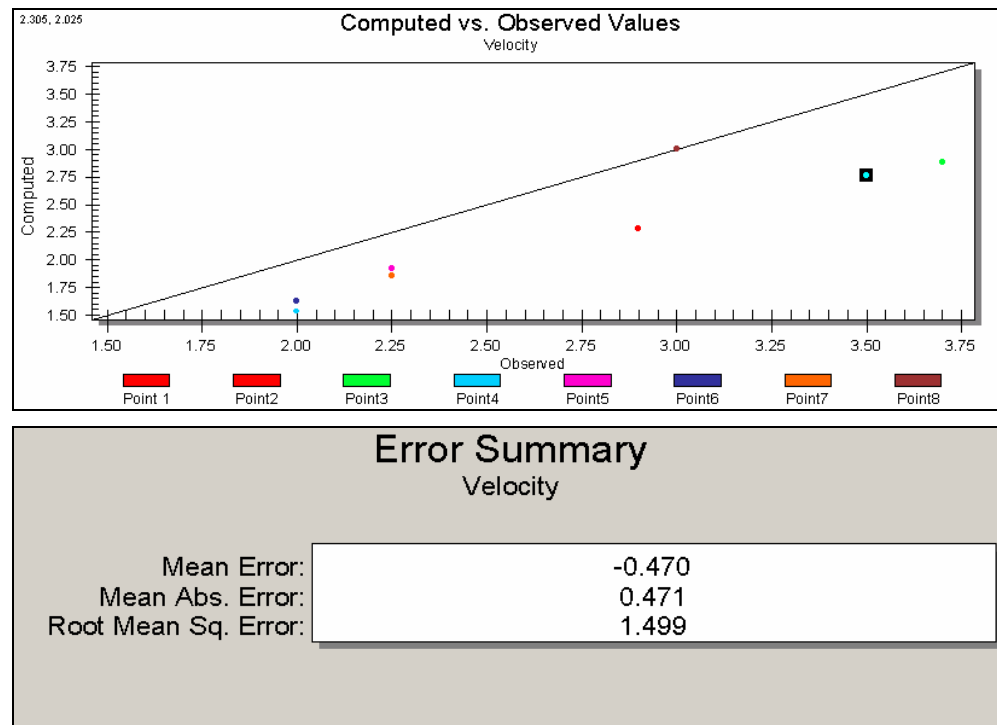


Figure 4-5 Computed vs. Observed Data and Error Summary plots

4.8.1 Plot Data

More plots can also be created for the *WSE* measurement or the current plots can be edited. To edit a plot:

1. Right-click the *Error Summary* plot and select *Plot Data...* from the menu.
2. Select *WSE* as the measurement.
3. Click *OK* to close the *Data Options* dialog.

The *Error Summary* plot is now updated using the data from the *WSE* measurement.

4.8.2 Using the Computed vs. Observed Data Plot

In the *Computed vs. Observed Data* plot, a symbol is drawn for each of the observation points. A point that plots on or near the diagonal line indicates a low error. Points far from the diagonal have a larger error. The position of the points relative to the line gives an indication whether the computed values are consistently higher or lower than the observed values. In this case, all points are below the line indicating that all computed velocities are lower than observed values.

Change the measurement for the *Computed vs. Observed Data* plot to the *WSE* measurement by following the steps above defined in section 4.8.1. Now, all points

plot above the line indicating that all computed water surface elevations are above the observed values.

4.8.3 Using the Error Summary Plot

In the *Error Summary* plot, the following three types of error norms are reported:

Mean Error. This is the average error for the points. This value can be misleading since positive and negative errors can cancel.

Mean Absolute Error. This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.

Root Mean Square. This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

Close the two plots now by clicking on the “X” at the upper right corner of the window. Use the Window|Tile command to enlarge the graphics window again.

4.9 Calibrating the Model

The values in this solution for both measurements are not within the calibration targets. To achieve better calibration, the material properties will be changed and then the model will be re-run. Since the errors through the main channel for the *Velocity* measurement are negative, indicating that the observed velocities are larger than those computed by the model, we want to change the parameters in such a way as to increase the velocity in these locations (eddy viscosity and/or Manning’s n). Increasing the velocity at these locations should also decrease the water surface elevation.

4.9.1 Editing the Material Properties

Decreasing the eddy viscosity values can increase these computed velocities. To decrease the eddy viscosity:

1. Click on the *Mesh* object to make it active.
2. Select *FESWMS | Material Properties...*
3. In the *Turbulence Parameters* tab, change the Vo (kinematic eddy viscosity value) from 10.0 to 1.5.
4. Click *OK* to close the *FESWMS Material Properties* dialog.

4.9.2 Computing a New Solution

To compute the new solution:

1. Go to *File* | *Save As...*, set the type to be a FESWMS project (.fpr) and save the simulation as *observe2.fpr*.
2. Run flo2dh on the new simulation. At the completion of the run, leave the toggle on for reading the solution.

4.9.3 Reading the New Solution

If you are using SMS in *Demo* mode, you will not be able to save the simulation. However, this second simulation has been saved in the *output* directory. You can open the second simulation by selecting the *File*|*Open* command and selecting the file “observe2.flo”.

The calibration targets will automatically update to show the errors for the solution that was just opened.

4.9.4 Fine-tuning the model

The verification targets now show that six points for the *Velocity* measurement are within the allowable range and two points are above the range, but still in the yellow range. There are no points more than the two times the variation above the observed value (red targets). Looking at targets for the *WSE* measurement four points are within the allowable range and four are below the range with one point being more than two times the variation below the observed value. Since the values for the *Velocity* measurement that are unacceptable are now higher than the observed values and the values for the *WSE* measurement that are unacceptable are now lower than the observed values, the correction made was too drastic. Specifically, the eddy viscosity was lowered too much and it needs to be raised. To compute another solution:

1. Click on the *Mesh* object to make it active.
2. Select *FESWMS* | *Material Properties...*
3. In the *Turbulence Parameters* tab, change the *Vo* (kinematic eddy viscosity value) from 1.5 to 6.0.
4. Click *OK* to close the *FESWMS Material Properties* dialog.
5. Save and run a third simulation of flo2dh (observe3.fpr). (Read in the simulation at the completion of the run or read the solution from the output folder.)

After this third simulation is opened, all the observation point targets for the *Velocity* measurement should be within the acceptable intervals. Make sure velocity is the active measurement and confirm this.

Now make the *WSE* measurement the *Active* measurement. All but two points are within the acceptable interval. To get these two points within range, compute another solution:

1. Change the kinematic eddy viscosity value (*Vo*) to 7.0.
2. Save and run a fourth simulation (observe4.fpr). (Read in the simulation at the completion of the run or read the solution from the output folder.)

Looking at the calibration targets should show all the points are acceptable. The calibration process is now complete. It will not always be possible to get all observation points for each measurement to be within the acceptable confidence interval. Therefore, it will have to be decided which measurements and which points are the most important to have within the acceptable range.

4.10 Using the Error Vs. Simulation Plot

When performing trial-and-error verification, it is often important to keep track of the error trend as new solutions are repeatedly computed. *SMS* provides a special verification plot to simplify this task. To create this plot:

1. Select *Display | Plot Wizard...* .
2. Choose *Error vs. Simulation* as the *Plot Type* and click *Next*.
3. Select *Velocity* as the measurement.
4. *SMS* will create a plot with one point for each simulation. The order of the points in the plot will follow the order solution sets in the *Solutions* list box. The solution at the top will be first. Use the *Move Up* and *Move Down* buttons to change this order. The default order is the order that they were read in.
5. Click *Finish* to close the *Plot Wizard* and generate the plot.

A new plot appears showing the *Error vs. Simulation*, as shown in Figure 4-6. Notice for the *Velocity* measurement that the errors decrease as each simulation was performed until the final solution where the errors slightly increase. This slight increase in error with the *Velocity* measurement was required to get that last observation point for the *WSE* measurement within the acceptable range. Generally, if the errors increase, then the model is not improving.

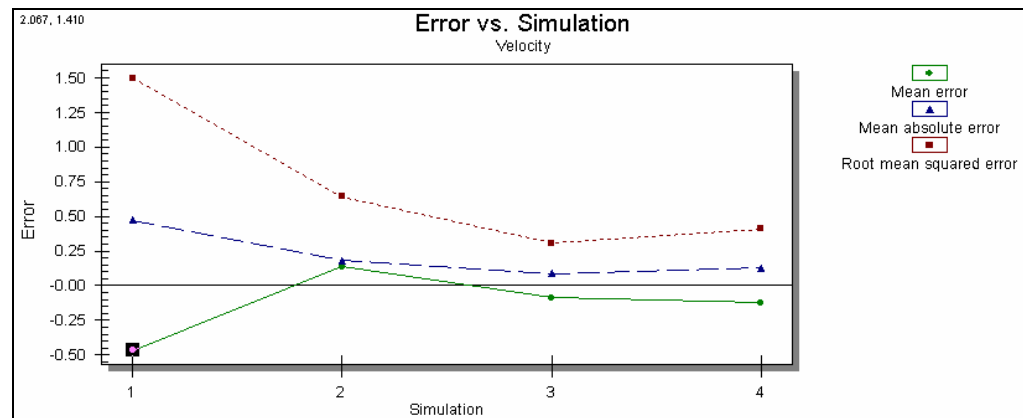


Figure 4-6 Error vs. Simulation plot for Velocity

Create another *Error vs. Simulation* plot using the WSE measurement. The errors for this measurement changed dramatically from solution to solution since parameters were first changed to calibrate points for the *Velocity* measurement. However, the general trend was a decrease in error. This plot is shown in Figure 4-7.

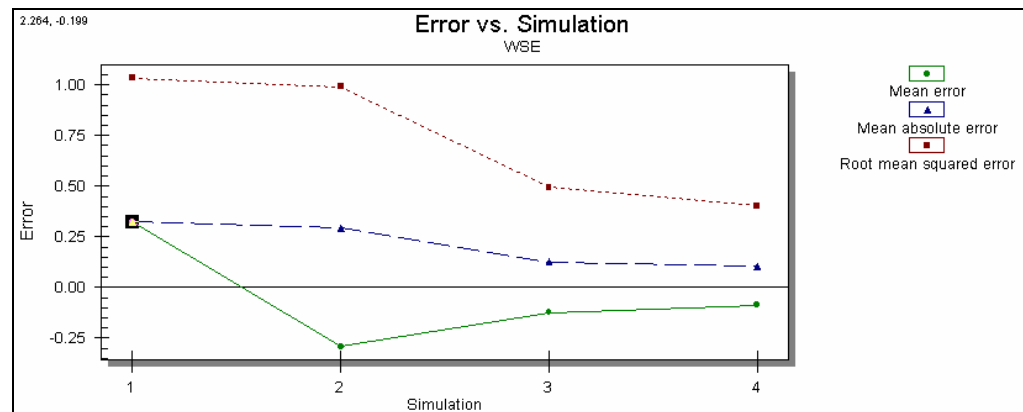


Figure 4-7 Error vs. Simulation plot for WSE

Close the plots now by clicking on the “X” at the upper right corner of the window.

4.11 Generating Observation Profile Plots

Observation profile plots are used to view data set values along observation arcs. The first observation arc to be created will be used to create a profile of the main channel. To create this arc:

1. Right click on the *Map Data* object in the *Project Explorer*. and create a new coverage. Click on this new coverage to make the Map module active. Change the coverage type to an observation coverage and change its name to “Profiles.” (We create a separate coverage to keep the observation arcs separate from the exiting observation points. When an observation arc is

being created, observation points may be clicked joining them to the arc. Observation points and arcs can exist on the same coverage.)

2. Choose the *Create Feature Arc*  tool from the *Toolbox*.
3. Create an arc down the main channel, as shown in Figure 4-8. Remember to double-click the last point to end the arc.

When the plots are drawn, they will use the name and color associated with the observation arc. To change the name and color of the arc:

1. Choose the *Select Feature Arcs*  tool from the *Toolbox*.
2. Double-click on the profile arc.
3. In the *Observation Coverage* dialog, change the *Name* of the arc to “river profile” in the *Observation Arcs* (leave its *Color* as the default).
4. Click *OK* to close the *Observation Coverage* dialog.

Three more arcs need to be created, each across a section of the river. These arcs will be used to create cross section plots. To create these arcs:

1. Select the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create each of the cross section arcs, as shown in Figure 4-8. Note: When creating these, DO NOT click ON the profile arc, as this would split it.

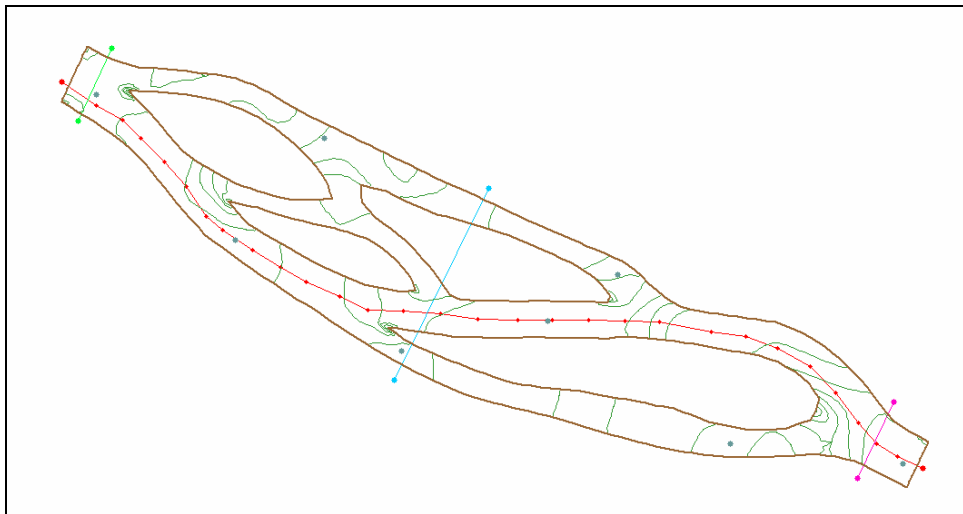


Figure 4-8 Profile and cross-section arcs

With the cross sections created, open the *Observation Coverage* dialog and assign a unique color and an appropriate name to each arc.

With the arcs created, the plots can now be generated. To do this:

1. Select *Display | Plot Wizard...* .
2. Choose *Observation Profile* as the *Plot Type* and click *Next*.
3. Turn on the *Use selected data sets* option, and check only the *elevation* data set in the *Generic Solution* and the *water surface* data set in the *observe4.flo* solution.
4. Turn off the three cross-section arcs in the *Arcs* spreadsheet by turning off their corresponding *Show* toggles.
5. Click *Finish* to close the *Plot Wizard* and generate the plot.

The profile plot of the geometry of the stream should appear as shown in Figure 4-9.

To view the velocity distribution across the three cross sections create a new *Observation Profile* plot. Turn on the *velocity mag* data set in the *observe4.flo* solution, and *Show* only the three cross section arcs.

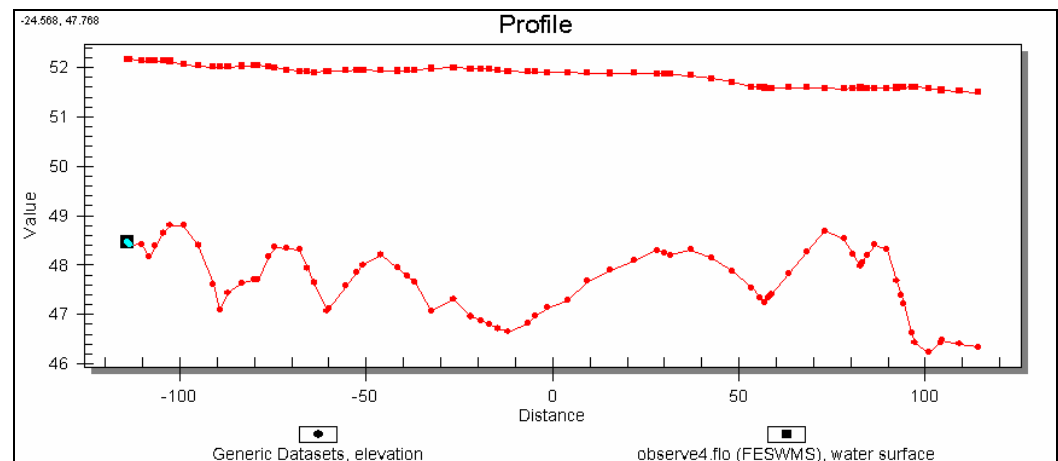


Figure 4-9 Observation Profile Plot

4.12 Generating Time Series Plots

As mentioned earlier, observation arcs are used to compute fluxes. One flux value that is often observed and measured in the field is flow rate. Observed flow rates can be used in model calibration in the same way observed velocities and water surface elevations are used. In addition to normal model calibration, *Time Series* plots can be generated showing how the flow rate flux changes with time. This type of time series plot is commonly known as a hydrograph. Hydrographs created using calculated data from the model are useful to see if the model properly predicts flow rate patterns. To create a *Time Series* plot:

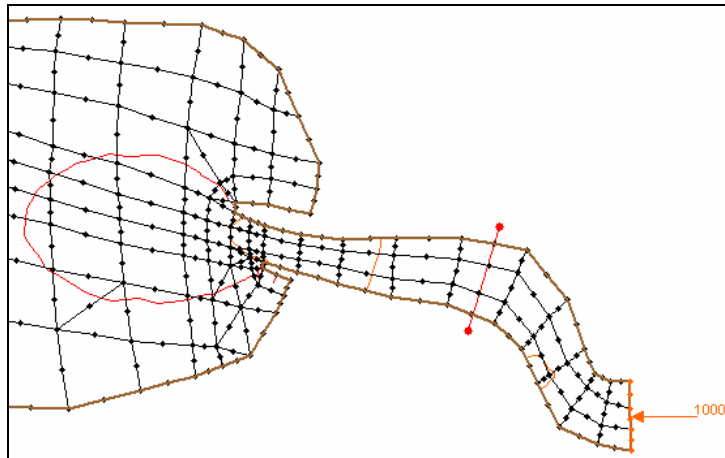


Figure 4-10 Observation arc across noyo1.spr mesh

1. Open the file *noyo1.sms* from the *tut04* directory. If geometry data is still open, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.
2. Create a new observation coverage called “Fluxes.”
3. Create an observation arc across the mesh as shown in Figure 4-10.
4. Open the *Plot Wizard* .
5. Choose *Time Series* as the *Plot Type* and click *Next*.
6. Change the *Function Type* to *Flux*. Check the *Show* toggle in the spreadsheet for the arc you just created (the default name will be “Arc #”). Select “water depth” as the *Scalar Dataset* and “velocity” as the *Vector Data Set*.
7. Click *Finish* to close the *Plot Wizard* and generate the plot.

A new window opens with the *Time Series* plot of the *Flow Rate* flux measurement. This plot should appear similar to the one in Figure 4-11.

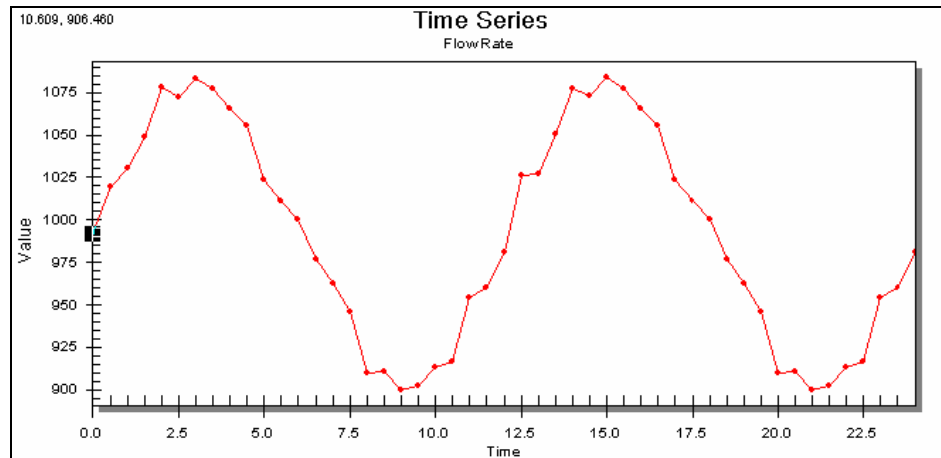


Figure 4-11 Time Series plot of Flow Rate.

4.13 Conclusion

This concludes the *Observation Coverage* tutorial. You may continue to experiment with the program or you can exit SMS. To quit SMS at this point:

- Select *File | Exit*. You may be asked if you want to save changes to your data. You don't need to do this.

Sensitivity Analysis

5.1 Introduction

This lesson analyzes the effects of changes in Manning's roughness coefficients and of kinematic eddy viscosity on various channel arrangements. Understanding the effects of Manning's roughness and eddy viscosity are useful in model calibration.

Either *RMA2* or *FESWMS* may be used for this lesson.

5.2 Simple Channel With Single Material

A flume 800 meters by 100 meters has been prepared for use in this lesson. The flowrate is set at 100 m³/s. The downstream water surface elevation is 1 m. This flume has no slope and is comprised of a single material.

5.2.1 Open The Simulation

To open the file that contains the necessary mesh.

1. Select *File* | *Open*.
1. Use the *tut05\rma2* directory if you are using *RMA2* and the *tut05\feswms* directory for *FESWMS*. Open *flumea1.sms*. If geometry data is still open from a previous tutorial, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.

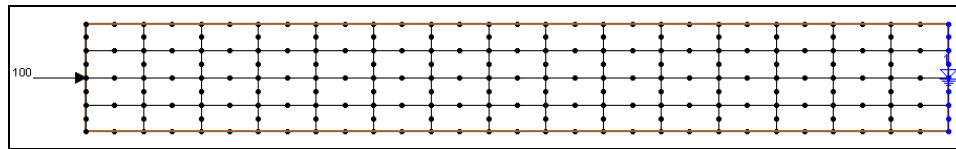


Figure 5-1 The mesh flumea1.

5.2.2 Running The Model

The correct material properties have been set for the initial run. You will need to run the model with the current settings. For instructions on running RMA2, see the Basic RMA2 Analysis. For instructions on how to run FESWMS, see the Basic FESWMS Analysis.

5.2.3 Creating Profile Plots

Before making a profile plot it is necessary to create an observation coverage, with an observation arc to define the profile to plot. To create an observation coverage and profile arc:

1. Select *default coverage* to activate it.
2. Right click on *default coverage*, go to type, and change the *coverage type* to *Observation*.
3. Select the *Create Feature Arc*  tool from the toolbox.
4. Create an arc down the center of the flume as shown in Figure 5-2

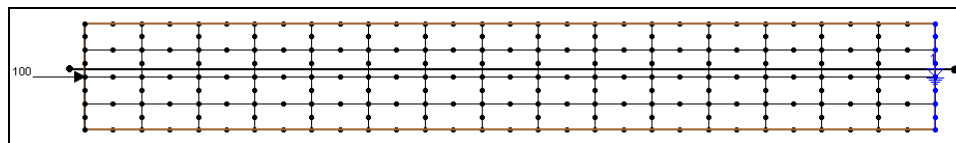


Figure 5-2 Mesh with Observation Arc.

In SMS, profile plots can be created to visualize the results of a model run. To create a profile plot:

1. Select *Display | Plot Wizard*.
2. Select the *Observation Profile* option and click *Next*.
3. Select the *Use selected data sets* option and then select “water depth”. Click the *Finish* button. The plot should now appear on your screen.

5.2.4 Varying Manning's Roughness

The next step is to change the material properties and rerun the model in order to compare the results. To change the material properties:

1. Click on Mesh Data to switch to the Mesh Module .
2. If using *RMA2*, select *RMA2 | Material Properties*. If using *FESWMS*, select *FESWMS | Material Properties*.
3. Change the *material_01* Manning's *n* (*n1* and *n2* for *FESWMS*) value to 0.045.
4. Select *File | Save As*.
5. Save the new project as *flumea2.sms*.
6. Rerun the model with the new roughness value, and read in the new solution file.
7. Right click on the plot and select Plot Data.
8. Turn on the newly computed water depth curve to add it to the plot.

Repeat steps 2-8 except change the *n* value to 0.065, and save the file as *flumea3.sms*.

The plot should now look like Figure 5-3. The plot demonstrates the fact that as the roughness increases, the upstream water surface elevation increases.

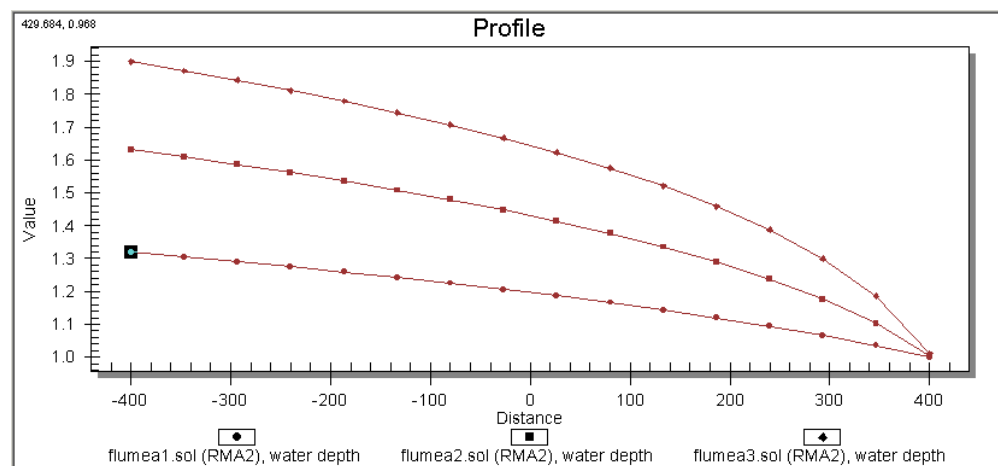


Figure 5-3 Water depth with varied roughness parameters for.

5.2.5 Changes in Eddy Viscosity

Eddy Viscosity is another parameter that can be modified to alter the model's solution. This section will analyze the effects of various eddy viscosities while keeping Manning's coefficient constant. To setup the first run:

1. First erase the old solutions by right clicking on them and choosing *Delete* .

2. If using *RMA2*, select *RMA2 | Material Properties*.

If using *FESWMS*, select *FESWMS | Material Properties*.

3. Change the *material_01* Manning's *n* value to 0.035.

4. If using *RMA2*, change the *Eddy Viscosity* (E) to 5.0.

If using *FESWMS*, change the *Viscosity* (Vo) to 1.0 m²/s.

5. Select *File | Save As*.

6. Save the project as *flumeb1.sms*.

7. Rerun the model with the new model parameters.

Now create two new solutions using steps 1-7. For *FESWMS* use viscosities of 10 and 100 m²/s. For *RMA2* use viscosities of 100 and 500,000. Name the files as *flumeb2.sms* and *flumeb3.sms*. Edit the plot data to show the water depths for these new runs instead of the previous runs.

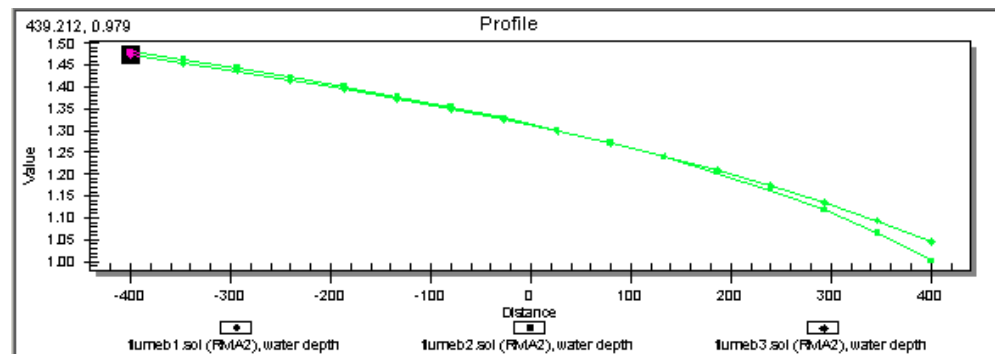


Figure 5-4 Eddy Viscosities of 4, 100, 500000 with $n = 0.035$ using *RMA2*.

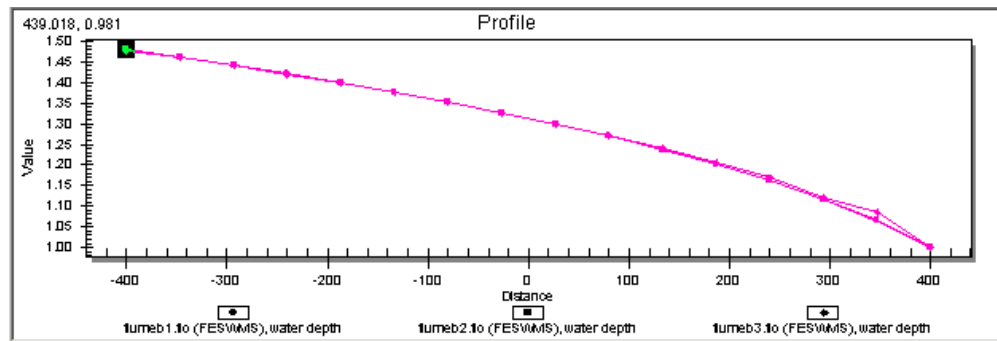


Figure 5-5 Eddy Viscosities of 1, 10, 100 m^2/s using FESWMS.

The results in *RMA2* changed little even for the unrealistic value of 500,000. *FESWMS* had no discernable difference even for values as high as 100 m^2/s .

5.3 Constrained Flume With Single Material

The second channel was designed to show the effect of roughness coefficients and eddy viscosities when large velocity gradients occur in the longitudinal flow direction. This channel has the same dimensions as our first flume, but it is constricted to 20 m wide through the middle. The channel has gradual contractions and expansions above and below the constricted section. The flowrate will remain 100 m^3/s . The downstream water surface elevation will remain 1 m.

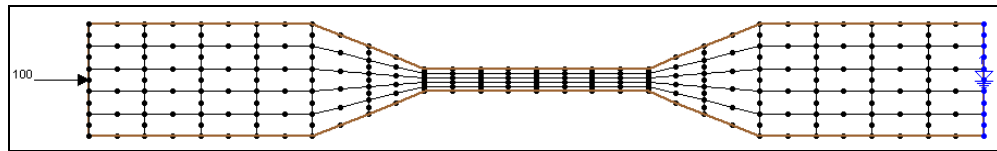


Figure 5-6 Test Channel #2

5.3.1 Open The Simulation

To open the new mesh:

1. First Select *File* | *Delete All*.
2. Select *File* | *Open* .
3. Select the file *flumec1.sms*.

5.3.2 Varying Manning's Coefficient

Repeat the same procedure as was outlined in sections 5.2.2 to 2. First, run the model as configured. Use n values of 0.045 and 0.065 for the subsequent model

runs. Save the files as *flumec2.sms* and *flumec3.sms*. Make sure to add all three solutions into the profile plot. When finished, the plot should look like Figure 5-7.

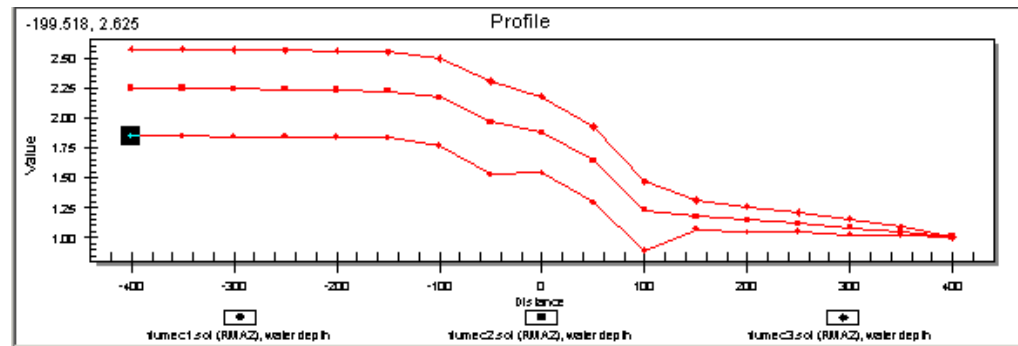


Figure 5-7 Constricted flume with various roughness factors.

5.3.3 Varying Eddy Viscosities

To analyze the effect of changing eddy viscosities:

1. In the *Material Properties* dialog change Manning's n value to 0.035.
2. If using *RMA2*, make sure the *Eddy Viscosity (E)* is 5.
If using *FESWMS*, make sure the *Viscosity (Vo)* is 1 m²/s.
3. Select *File | Save As*.
4. Save the file as *flumed1.sms*.
5. Repeat sections 5.2.2 to rerun the model with the new model parameters.

Now create two new solutions using steps outlined above. For *FESWMS* use viscosities of 10 and 100 m²/s. For *RMA2* use viscosities of 1000 and 10,000. Name the files *flumed2.sms* and *flumed3.sms* respectively. Import the new solutions into the *Data Browser* and turn on the water depth functions in *Plot Data* dialog.

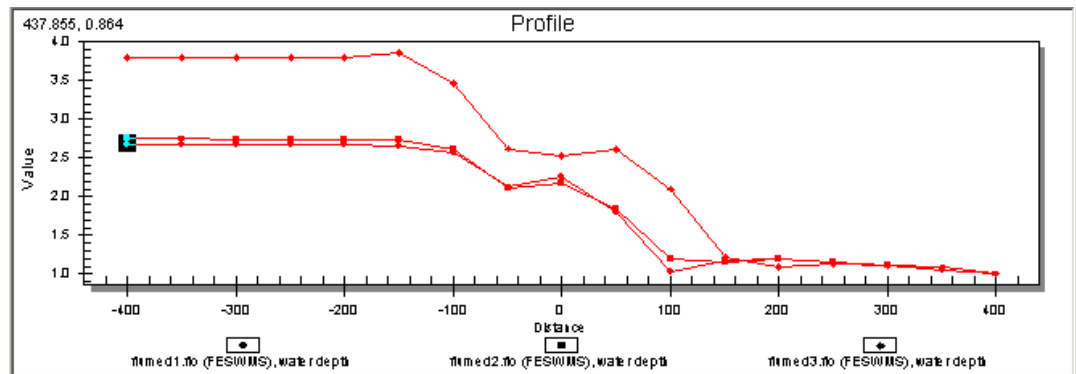


Figure 5-8 FESWMS run of the constricted flume at various eddy viscosities

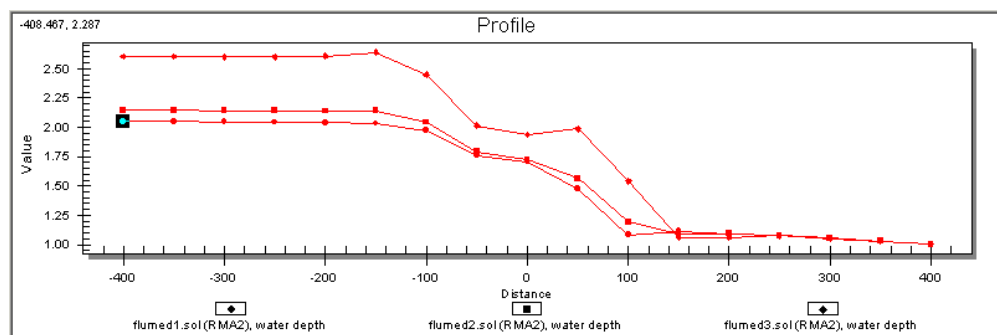


Figure 5-9 RMA2 run of the constricted flume at various eddy viscosities.

As shown in Figure 5-8 and Figure 5-9, eddy viscosities have a much larger effect when there are large longitudinal velocity gradients. For realistic values of eddy viscosity, differences in depth at the upstream end of the channel are small.

5.4 Simple Channel With Two Materials

This channel has the same dimensions and boundary conditions as the first one. The elements are smaller and the channel has two material types rather than one. We will examine the effects the lateral roughness variation has on velocity.

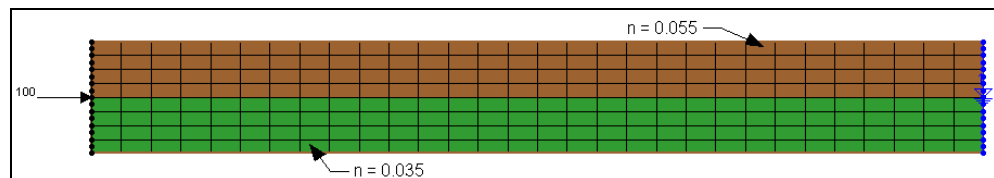


Figure 5-10 Channel #3 simple flume with two materials

This time specific instructions will not be given. If you don't remember how to do something, look back at the lesson for help.

1. Open the file *flumee1.sms*.
2. Run the simulation with the current settings.
3. For *RMA2* rerun the model with viscosities of 500, 5000, and 50,000 for both materials.

For *FESWMS* rerun the model with viscosities of 5, 50, and 100 for both materials.

Name the simulation files *flumee2.sms*, *flumee3.sms*, and *flumee4.sms*.

4. Create an observation coverage.
5. Create an observation arc across the flume at about 200 m from the downstream boundary as shown in Figure 5-11.
6. Create an observation profile plot turning on the velocity mag function for each solution.

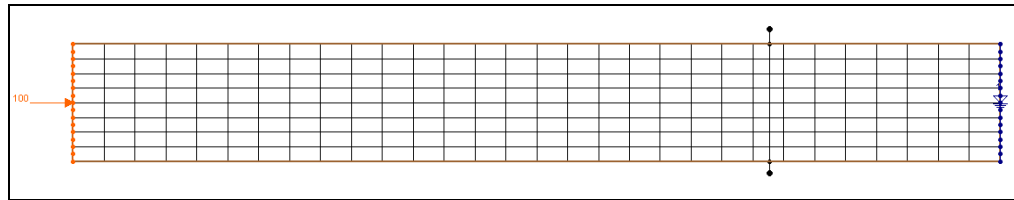


Figure 5-11 Channel #3 showing placement of observation arc.

The plot should look like Figure 5-12 or Figure 5-13. As you can see in the graph, smaller eddy viscosities allow larger transverse velocity gradients to appear in the solution.

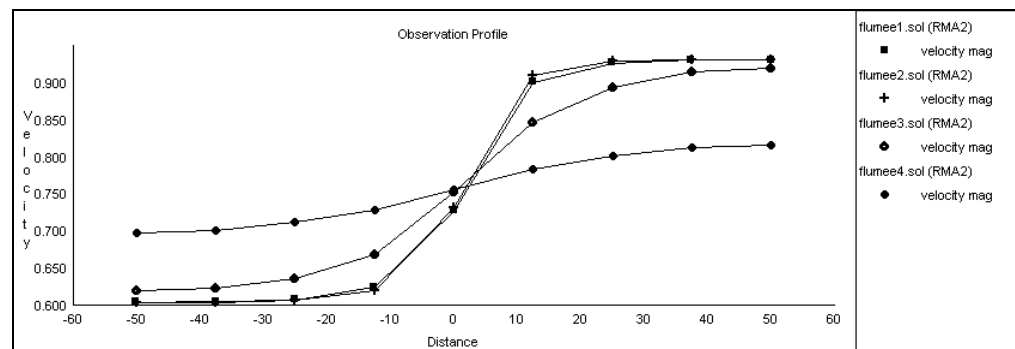


Figure 5-12 Profile plot of RMA2 solution for various eddy viscosities for channel #3.

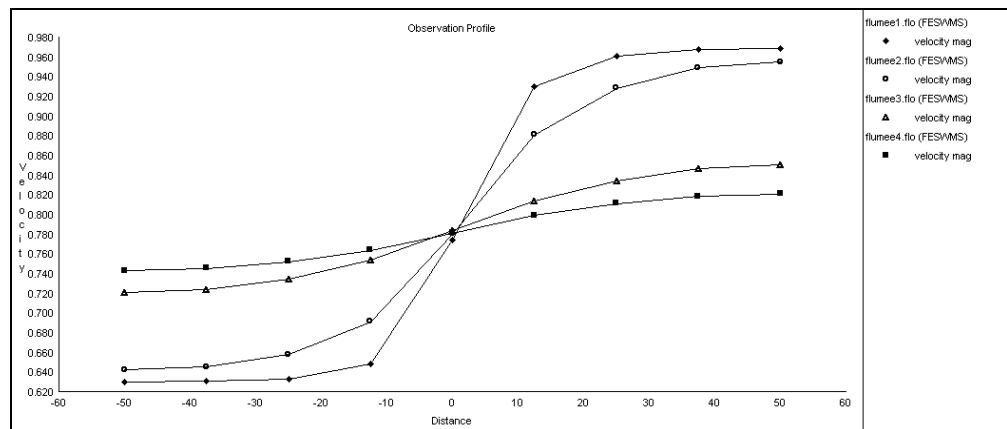


Figure 5-13 Profile plot of FESWMS solution for various eddy viscosities for channel #3.

5.5 Conclusion

This concludes the *Sensitivity Analysis* tutorial. You may wish to experiment further with different channel arrangements and watch the effects of changing roughness and viscosity values. This concludes the tutorial. If you wish to exit *SMS* at this point:

- Choose *File* | *Exit*.

Generic 2D Mesh Model (Gen2DM)

6.1 Introduction

This lesson will teach you how to use the generic mesh model as an interface for models not executed within *SMS*. *SMS* creates a file, with the extension “*.2dm,” which includes mesh, parameter, boundary condition and material data that may be used by other executables. The process of generating this file involves defining the model within *SMS*, using the definition and a mesh to represent a situation, and finally saving the file to be imported elsewhere. All information defined and used within *SMS* for a specific outside model remains with the file, therefore the creation of a model definition master “*.2dm” (a Gen2DM file with no mesh and variable assignments; it contains only information pertaining to the model definition) is recommended. The master file can then be opened as an established beginning, added to and saved as a new project. To begin this lesson, define the example model interface and save it, then utilize the file for an example project.

To manually change to this model, select the Mesh Module icon  or select the *Mesh Data* parent in the Project Explorer (if mesh data currently exists), then select *Data | Switch Current Model*. Select *Generic* and click the *OK* button. The Generic model can be recognized by the *Mesh* menu. This menu name will soon change though, when defining the model interface.

6.2 Specifying Model Units

Before continuing, make sure that the units are as desired. To do this:

1. Select *Edit | Current Coordinates*.
2. Make sure the *Horizontal* and *Vertical System* and *Units* are appropriate for the model executable outside of SMS. For this example, set *Horizontal System* to *State Plane NAD 83 (US)*, *Units* to *U.S. Survey Feet*, *St. Plane Zone* to *Pennsylvania South-3702*, *Vertical System* to *Local* and *Vertical Units* set to *U.S. Survey Feet*.
3. Click *OK* to exit the dialog.
4. Select *Mesh | Define Model*
5. Click the *Global Parameter Definition* button
6. For *Time Units*, enter the desired unit the model will be using. The unit will not be used in SMS, but it will be written to the file for reference. Enter “minutes.”

6.3 Defining the Model Interface

Model interface parameters will define various states and characteristics of a model. These model parameters may include items such as those needed to describe flow, channel roughness, and control structures. Depending on the intentions and capabilities of the outside executable, parameters will be organized into groups and given suitable value ranges. Proper organization of parameters will increase the abilities of SMS as an interface, especially for executables that are designed for multiple simulation possibilities.

6.3.1 Global Parameters

Before proceeding farther, the interface should be given the name of the model SMS is not currently integrated with. Under *Model Information*, of the *Mesh Global Parameters* dialog, enter “Gen2DM” (for this example). Upon exiting this and the main *Define Model* dialog, the menu item previously titled *Mesh* is now *Gen2DM*. Other dialogs specific to this model interface will also have titles including our entered name. Similar to the time unit variable mention above, the model name is written to the file and used as a reference. By giving the model interface the model name, files can be opened in SMS and quickly recognized as pertaining to a particular model.

To define the model parameters, in the same dialog, first create a parameter group:

1. Click in the *Parameter Groups* spreadsheet and type the name “Hydrodynamic.”
2. Click the *Define* button which was just enabled. This will open the *Hydrodynamic Parameter Definition* spreadsheet.
3. Enter the first *Hydrodynamic* parameter, “Time interval”. When editing the name is complete, the accompanying fields will be enabled.

4. Select *integer* in the *Type* column.
5. Set the *Default* to be 20 minutes (the time unit was previously declared as minutes).
6. The fictitious model will accept only positive time intervals, consequently declare the *Minimum* for *Time interval* as zero and leave the *Maximum* as blank .
7. Continue adding the following parameters with their limitations:
 - “Velocity max (ft/sec),” type: real number, default of 75.0, range of 0.0 to 100.0
 - “H min (ft) ,” real number, 0.25, positive
 - “A min”, real number, 1.0e-15 (entered just as appears), no range (leave both bounds blank)
 - “Check for dry elements”, boolean, on (checked)
 - “Element style”, text, “quadratic”
 - “Critical scour velocity”, options, “2.0 ft/sec”, defined options of 0.8 ft/sec, 2.0 ft/sec, 2.6 ft/sec (Click the *Define opts* button and enter options before selecting the default option selection).
 - “Average density (slug/ft³)”, real number, 3.0, 1.5 to 6.0
8. Click *OK* to save the definitions

Follow the same procedure to declare parameters for sediment transport, but first remove the *Average density* parameter from the *Hydrodynamic* group. It is a disparate parameter compared to this group. Enter the *Hydrodynamic Parameter Definition* spreadsheet and select the name cell and push the delete key or edit the name and delete all characters within the cell.

- ❖ Note: All spreadsheets in *Generic Model Define* that require names for groups, parameters, options, etc. necessitate unique names for that spreadsheet. Different parameter groups may have parameters with the same name, for example, but each group cannot. Also, deleting entire definition rows in these spreadsheets always consists of deleting the name.

Repeat steps 1-8 above for the “Sediment transport” parameter group with the following parameters:

- “Time interval,” integer, 10, positive
- “Source X position,” real number, 0.0, no range

- “Source Y position,” real number, 0.0, no range
- “Source elevation,” real number, 0.0, no range
- “Parcel mass (slug),” real number, 0.5, greater than 0.0001
- “Particle mass (slug),” real number, 0.003, greater than 0.0001
- “Particle size (in),” real number, 0.05, positive
- “Deviation,” real number, 0.0, no range
- “Average density (slug/ft³),” real number, 3.0, 1.5 to 6.0

You should now have two complete parameter groups defined for later use. Click *OK* to save all data appertaining to global parameters of the model interface. If you had chosen *Cancel*, all data would have been lost, even the individual parameters. You should be at the *Define Model* dialog.

6.3.2 Boundary Conditions

A mesh and a model are insignificant without circumstances to simulate water flow. Gen2DM (the SMS interface) allows boundary conditions to be specified for three entities: nodes, nodestrings, and elements (linear and quadratic triangles and quadrilaterals). The conditions may be defined for general use, or correlated with a particular parameter group and hence its availability limited. To define a boundary condition:

1. Click the *Boundary Condition Definition* button.
2. On the *Node* tab, enter the name, “Water sink/source.”
3. Change ID to 2.
4. Leave *Legal on interior* checked (This refers to whether this condition can be assigned within the mesh, in addition to along the mesh boundary).
5. Click the *Define* button to enter the *Water sink/source Definition* spreadsheet.
6. Add the value “Flow rate (cfs)” with a default value of 0.0 and positive bounds.
7. Add “Water temperature (F),” 65.0, 32.5 to 100
8. Click *OK* to save and exit the spreadsheet.
9. Continue adding the following boundary conditions and their parameters and limitations:

a. *Node* tab:

- “Ceiling (pressure flow),” not legal, value: “Ceiling (ft above sea level),” 0.0, no bounds
- “Water surface observation gauge,” legal, no defined values

❖ Note: The IDs of boundary conditions can be reassigned as long as they are not duplicated. The default for a new ID is the lowest unused integer. Also, conditions may be comprised of none or multiple values depending of the model executable’s design.

b. *Nodestring* tab:

- “Water surface,” not legal, values: “Elevation,” 0.0, no bounds; “Essential/Natural factor,” 0.0, 0.0 to 1.0; “Vary along nodestring factor,” 1.0, 0.0 to 10.0;
- “Flow,” not legal, value: “Flow rate (cfs),” 0.0, positive
- “Supercritical,” not legal, no defined values
- “1D weir segment,” legal, values: “Discharge coefficient,” 1.0, positive; “Weir width (ft),” 1.0, positive; “Crest level (ft above sea level),” 0.0, no bounds; “Equation (0 = water level / 1 = energy head),” 0.0, 0.0 to 1.0
- “Sediment trap,” legal, no defined values

c. *Element* tab:

- “2D weir,” not legal, values: “Discharge coefficient,” 1.0, positive; “Crest level (ft above sea level),” 0.0, no bounds; “Equation (0 = water level / 1 = energy head),” 0.0, 0.0 to 1.0

The boundary conditions have been declared now for all entities to be utilized, however the nodestring boundary condition *Sediment trap* is only needed when a simulation depicts sediment transportation.

10. Check the *Specify parameter group correlation for boundary conditions* toggle and the bottom of the *Gen2DM Boundary Conditions* dialog. This enables a new column in each entity spreadsheet.
11. Select the *Nodestring* tab.
12. For *Sediment trap*, under *Corr. Param. Group* (Correlated Parameter Group), select *Sediment transport*. Leave all other conditions as (*none*) to allow generality.

- ❖ Note: If the *Specify parameter group...* toggle is unchecked, all correlations will be removed and upon rechecking the toggle, correlations must be reset if so desired. The parameter groups available to choose from are only those defined in *Global Parameter Definition* and saved.

13. Click *OK* to save and exit this dialog.

The remaining portion of the model interface to define is material attributes.

6.3.3 Material Properties

The *Material Property Definition* button is used to set the attributes of the simulation's mesh. Akin to a simulation without boundary conditions, a bathymetry without friction is senseless. To define mesh property parameters:

1. Click the *Material Property Definition* button.
2. Click in the *Values* spreadsheet and type the name "Manning."
3. Set the *Default*, *Minimum*, and *Maximum* to 0.035, 0.01, and 0.18, respectively.
4. Continue by adding the following parameter and its characteristics:
 - "Kinematic eddy viscosity," 0.0, no bounds
5. Click *OK* to save and exit this dialog.

The master "*.2dm" file is now complete for this model interface.

6.4 Protecting and Saving the Model Definition

It is recommended to protect the model interface from accidental manipulation once it has been defined and to always have a version ready for new simulations. By saving a clean master ".2dm" for each outside model executable, SMS capabilities are enhanced and more efficient for the user. To guard and save the *Gen2DM* definition just created:

1. In *Define Model*, check the *Lock model definition* toggle.
2. Enter the password "sms-gen2dm" in the enabled *Key* field.
3. Click *Close* to finish.
4. Return to the dialog by selecting *Gen2DM | Define Model*.
5. The *Definition Is Locked* window will appear, requiring the password. Enter the key.

- ❖ Note: The key is case sensitive. If the key is incorrect, a message will appear and necessitate an *OK* click. The password is not protected or encrypted. It is written to the file and can be found easily by opening the file in a text editor. Please refer to section 24.11 Gen2DM (*.2dm) File Format for more information.
- 6. Upon reentry into the model definition, Click *Close* again.
- 7. Select *File | Delete All* (To make sure we do not have any mesh information before saving the master file).
- 8. Click *Yes* to keep the *Gen2DM* definition.
- 9. Click *Yes* to delete all other data.
- 10. Select *File | Save Gen2DM*.
- 11. Explore to the *tutorial\tut24* directory.
- 12. Enter “Master Gen2DM” as the *File name*. Verify that the file type is “2D Mesh Files (*.2dm).”
- 13. Click *Save*.

If a master file does not exist and a simulation does exist, open the file and perform steps 7-13 with step 10 selecting *Save As* instead to create one. To prevent the repetition of redefining the interface, always back up and store at least one copy of the initial master file or a simulation (“*.2dm” file with mesh and assignment information).

6.5 Assigning Model Parameters

For the remainder of this tutorial use the “*.2dm” file with the *Gen2DM* model definition just created and a geometry. To close the file and open the next:

1. Select *File | Delete All*.
2. Click the *No* button to the message. You want to delete all.
3. If another deletion message appears, click the *Yes* button.
4. Using your computer operating system, explore to the *tutorial\tut24* directory, select and drag the file “Conestoga_River.2dm” onto the canvas of *SMS* and drop it. If the directory window and *SMS* canvas are not visible together on the screen, first drag the file to the *SMS* icon on the taskbar which should cause *SMS* to come forward on the screen and then drop the file within the canvas. If you have mesh data open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

Or:

4. Select *File | Open*.
5. Open the file “Conestoga_River.2dm” from the *tutorial\tut24* directory. If you have mesh data open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The geometry data will open, as shown in

Figure 6-1. SMS will automatically be in the Generic model of the Mesh Module after the file is opened with the menu *Gen2DM*. You therefore know that this file is associated with the “Gen2DM” model executable outside of SMS (fictitious model executable; the name is only an example for this tutorial).

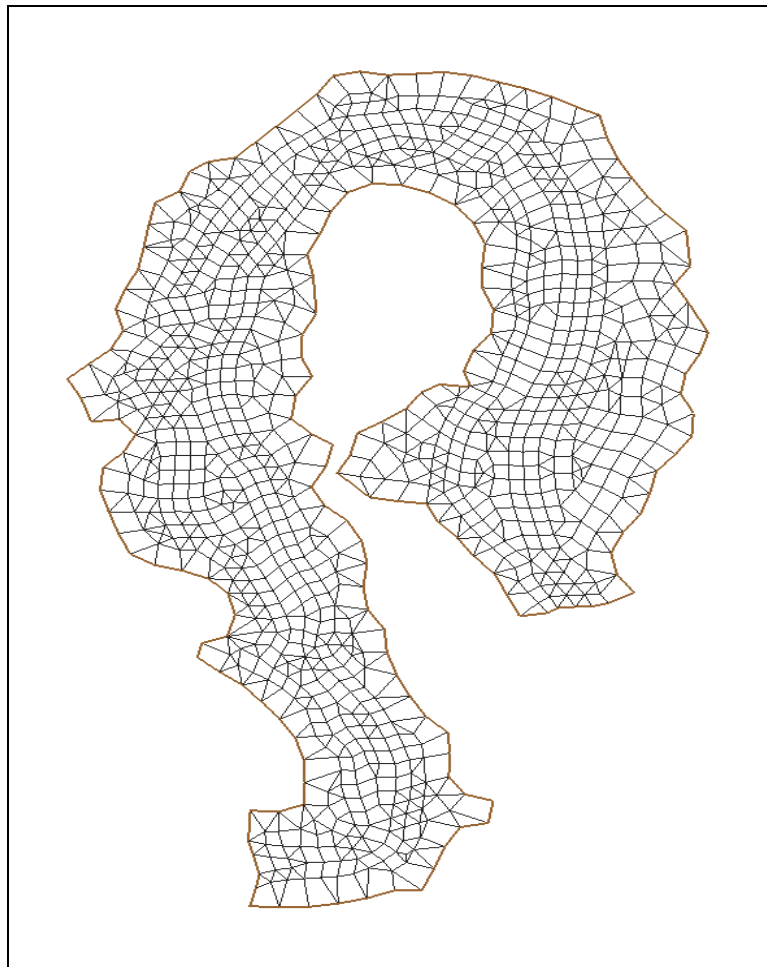


Figure 6-1. The mesh contained in the file *Conestoga_River.2dm*.

The Gen2DM file does not write out the coordinate system for later use, so redefine the system since you have performed a delete all data SMS command. Follow steps 1-3 in section 24.2 Specifying Model Units.

If you selected *Gen2DM | Define Model* and entered the password “sms-gen2dm,” the predefined model interface can be reviewed and edited. You will now be using this definition.

1. Select *Gen2DM | Global Parameters*.

- ❖ Note: This dialog contains three tabs; the first tab is for general parameters and the additional are for each parameter group defined in the model. The model is initialized to *Steady state*.

2. Select the *Hydrodynamic* tab.

- ❖ Note: All variables have been initialized to the defined default values and *Activate parameter group* is checked. Also any boundary conditions that are correlated to this group are listed. If (*none*) is stated, then no conditions of that entity are specific to this group. If entity boundary conditions are not represented, then the *Specify parameter group...* toggle in *Gen2DM Boundary Conditions* definition is unchecked.

3. Enter the value -100 in the *Time interval* field.

4. The value should have returned to zero, because the acceptable range was defined as positive in the definition. Enter 150.50.

5. The value becomes 150, because the variable was declared an integer.

6. Enter other values and make selections as desired, testing the variable limitations.

7. Select the *Sediment transport* tab and likewise examine.

8. Select *OK* to exit.

Global parameter values may be changed at any time by accessing the *Gen2DM Global Parameters* dialog. Upon saving this file, all current values (including the default value if not changed) are written out for use by the model executable.

6.6 Appending a Mesh to a Current Defined Gen2DM

Manual copying and pasting of mesh data to a Gen2DM file (pasting information somewhere between the *MESH2D* and *BEGPARAMDEF* cards) is discouraged because mistakes involving the ids of entities may cause unforeseen problems. It is best to open additional data in *SMS*. To append in *SMS* with one “*.2dm” file already opened:

1. Select *File | Open*.

2. Highlight the Gen2DM file named “Conestoga_River_Addition.2dm”.

3. Click the *Open* button.

4. Click the *Append* button.
5. Click *Yes*.

The additional mesh data will be added to the current mesh. When joining Gen2DM files, SMS will only read the mesh data. The model definition and assignments of the current file will not be appended to. In fact, the boundary condition assignments and any existing datasets (including the elevation scalar function) will be deleted do to the bathymetry change. It is recommended that all appending be completed before assigning conditions. Save the current file as “Conestoga_River_Combined.2dm” to keep the supplementary mesh included.

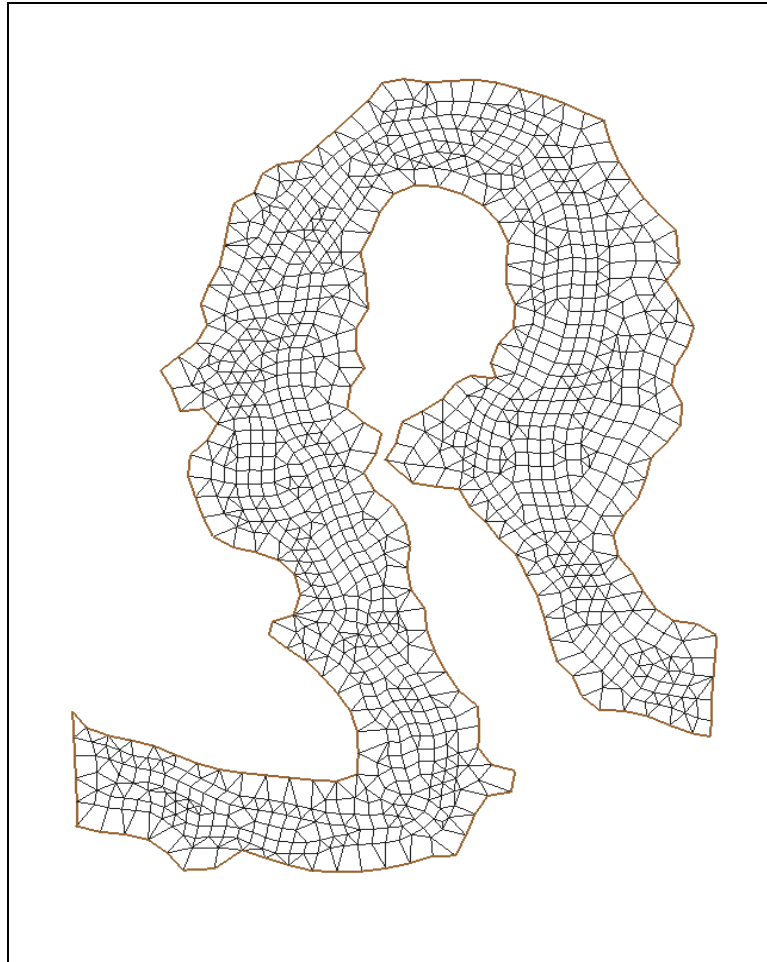


Figure 6-2. The mesh contained in the file *Conestoga_River.2dm* after appending.

6.7 Assigning Boundary Conditions

Before assigning boundary conditions, ensure the mesh composition complies with the outside SMS executable. Some models may only support certain element forms

(triangular/quadratic) or advise against various mesh complexities. Since our model is conjured just as an example, the mesh will be assumed to be compliant. For adjustment of a mesh, use the options available in the *Nodes*, *Nodestrings* and *Elements* menus.

6.7.1 Creating Nodestrings

For most simulations, boundary conditions will be declared along nodestrings at the *open boundaries* of the mesh. An open boundary is a boundary where water is allowed to enter or exit. Generally, a flow rate is specified across inflow boundaries and water surface elevation is specified across outflow boundaries for a very simplistic run.

This example will have one inflow boundary and one outflow boundary so two nodestrings must be created. These boundaries are highlighted in Figure 6-33.

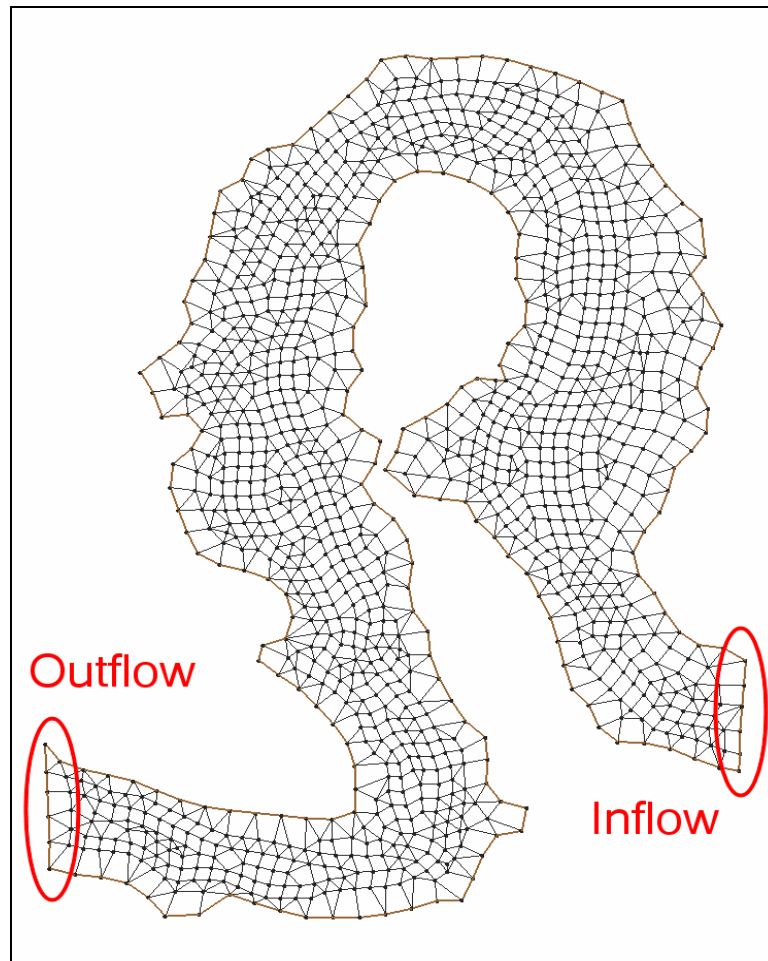


Figure 6-3. Position of the boundary nodestrings in the mesh.

Nodestrings should be created from right to left when looking downstream and both nodestrings should span the entire river section. It does not matter which nodestring is created first. To create the outflow nodestring:

1. Choose the *Create Nodestrings* tool  from the *Toolbox*.
2. Start the nodestring by clicking on the upper node at the outflow boundary.
3. Hold the *SHIFT* key and double-click on the lower node at the outflow boundary to create and end the nodestring.

The inflow nodestring can be created likewise, making sure you create it right to left when looking downstream. Also create two nodestrings anywhere on the interior of the mesh (boundary nodes can be included in an interior nodestring as long as at least one non-boundary node is incorporated).

6.7.2 Assigning Boundary Conditions

To assign a boundary condition, choose the selection tool for an entity and select a desired entity. Assign conditions to the nodestrings, a node and an element:

Nodestrings:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring.
2. Select the inflow nodestring (on the right side of mesh) by clicking on the icon.
3. Select *Gen2DM | Assign BC*.
4. Change the *Group* selection to *(none)*. This updates the options in *Type* and the spreadsheet.
- ❖ Note: The *(none)* group consists of all boundary conditions for this entity that are not correlated to a specific parameter group. Active parameter groups which contain conditions are listed first. If the *Specify parameter group...* toggle in *Gen2DM Boundary Conditions* definition is unchecked, then the *Group* selector will be disabled, allowing only for a *Type* selection.
5. Select *Flow* in *Type*. This updates the spreadsheet again.
6. Enter a *Flow rate* of 300.0 cfs.
7. Click *OK* to assign the boundary condition.
8. Select one of interior nodestring by clicking on the icon.
9. Select *Gen2DM | Assign BC*.
10. Change the *Group* selection to *(none)*.
11. Look at the options in *Type*.

- ❖ Note: The *Type* selector will only consist of conditions which have had the *Legal on interior* check box in *Gen2DM Boundary Conditions* entity definition spreadsheet checked if the entity to be given the assignment is not along the mesh boundary.

12. Click *OK* to assign the type *1D weir segment* using the default values.
13. Select the remaining interior nodestring and assign it the *Sediment trap* boundary condition (in the *Sediment transport* group).

Nodes:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. Select a group of interior nodes by clicking and dragging a selection box.
3. Select *Gen2DM | Assign BC*.
- ❖ Note: The *Group* selector is replaced with the text (*none*) because all conditions of this entity are global (not correlated to a parameter group).
4. Select *Water surface observation gauge* option in *Type*.
5. This condition does not contain any values to be entered, so select *OK* to assign the boundary condition to all selected nodes.

Elements:

1. Choose the *Select Elements*  tool from the *Toolbox*.
2. Select an interior element (not bordering the mesh boundary).
3. Select *Gen2DM | Assign BC*.
- ❖ Note: Different messages may appear before entering the *Gen2DM [Entity] Boundary Conditions* window with information about the model definition or the selected entity.
4. Click *OK* to this message.
5. Select an element along the inflow nodestring.
6. Select *Gen2DM | Assign BC*.
7. The window now appears, but select *Cancel* because the weir should be assigned inside the mesh boundary.
8. Select *Gen2DM | Define Model*.

9. Enter the key, “sms-gen2dm” into the field.
10. Click on the *Boundary Condition Definition* button.
11. Select the *Element*.
12. Check *Legal on interior* for *2D weir*.
13. Select *OK*.
14. Select the *Close* button.
15. Select an interior element (not bordering the mesh boundary) and assign it the *2D weir* boundary condition using default values.

6.7.3 Correlation and Activation Benefits of Boundary Conditions

The *Sediment transport* parameter group will not be used any further during this simulation assignment, so to simplify assigning the outflow boundary condition, turn off the group.

1. Select *Gen2DM | Global Parameters*.
2. Select the *Sediment transport* tab.
3. Uncheck the *Activate parameter group* check box.
4. Click *OK*.
5. Choose the *Select Nodestrings*  tool from the *Toolbox*.
6. Select the outflow nodestring (on the left side of mesh).
7. Select *Gen2DM | Assign BC*.
- ❖ Note: If all parameter groups which have boundary condition correlations are inactive, the *Group* selector is replaced with the text (*none*). If some groups with correlations are active and others inactive, the inactive boundary conditions will not appear in the *Group* selector.
8. Select *Water surface* in the *Type* selector.
9. Enter 80.0 for *Elevation*.
10. Click *OK* to assign the boundary condition.

If you selected the inflow nodestring again and reassign the boundary condition, condition type *Flow* will be displayed with a value of 300.0 cfs. If you selected the *Sediment trap* boundary condition type nodestring, a message will be shown informing

that the previously assigned data could not be found. The type is currently inactive because the *Sediment transport* group is inactive. The previous data can be left unchanged or deleted, whereupon you can reassign an active condition. To show an example of the boundary display options, leave the *Sediment trap* boundary condition type nodestring assigned as such.

6.7.4 Boundary Condition Display Options

You may have noticed that the entities with assigned conditions have symbols and labels, all of which are in black. To increase the visibility of certain assignments, change the attributes of each:

1. Click the *Display Options*  macro or select *Display | Display Options* or press *CTRL+d*.
2. The *2D Mesh* tab should be displayed on top, if not, select it.
- ❖ Note: The Generic 2D Mesh model is the only model in the mesh module which has the *Elements Options* button.
3. Click the *Nodestrings Options* button.
4. Every nodestring boundary condition defined in the model is represented in the spreadsheet. Click the middle of the line style button for *Water surface*.
5. Toggle *Solid* on and enter a width of 5.
6. Change the *Line Color* to red.
7. Click *OK*.
8. Repeat for *Flow*, but with bright green.
- ❖ Note: Gen2DM boundary condition nodestrings receive a hollow circle symbol displayed at their midpoints. The symbol receives the same color as the nodestring line.
9. Uncheck the toggle next to *1D weir segment*.
10. Change the *Inactive nodestrings* to be orange.
11. Change *Nodestring labels* to be blue by clicking on the down arrow beside the text preview.
12. Check *Show BC values in labels*.
13. Follow the same procedure and adjust the display options for *Nodal BC* and *Elements* to be clearly discernable.

❖ Note: Nodal BC symbols may be difficult to see if below the size of 3.

14. Uncheck *Nodal BC labels* in *Gen2DM Nodal BC Display Options*.

15. Click *OK* to close the *Display Options* dialog.

The check boxes on the *2D Mesh* tab of *Display Options* enable and disable the display of all instances of the entity. The check boxes next to each boundary condition type enable and disable each specific entity with that condition assignment. The *Inactive [entities]* and *[Entity] labels* check boxes are for all boundary conditions of that entity type. The attributes of *Inactive [entities]* is given to each instance of the entity that have a condition correlated in an inactive parameter group. Also, the label of an inactive entity is colored with the inactive color. Inactive nodestrings receive a second smaller hollow circle as part of their symbol, as shown in figure 6-4.

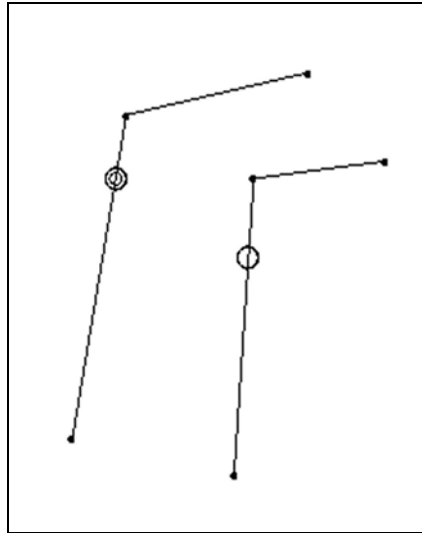


Figure 6-4. Symbols of inactive nodestrings (left) and active nodestrings.

6.7.5 Dynamic Boundary Conditions

The boundary conditions of the model may be defined dynamically to allow for varying conditions by creating a curve for each dynamic value. To describe the inflow nodestring of type *Flow* as changing flow rates:

1. Select *Gen2DM | Global Parameters*.
2. Toggle *Dynamic*.
3. Enter 20.0 and 1000.0 for *Time step* and *Total time*, respectively.
4. Click *OK*.

5. Select the inflow nodestring.
6. Select *Gen2DM | Assign BC*.
7. A new column entitled *Constant* should be available. Turn off the option.
8. Click the *Define* button.
9. Define the *Flow rate* curve in the *XY Series Editor* with the following values:
 - Time = 0.0, Value = 300.0
 - 20.0, 380
 - 40.0, 400
 - 60.0, 380
 - 80.0, 300
 - 100.0, 300
10. Click *OK*.
11. Click *OK*.

The data for each curve defined is stored by *SMS* by curve ID, but when written to file, each curve will be written out as a value for every time interval and given the entity and type ID it is describing.

6.7.6 Deleting Boundary Conditions

To delete a boundary condition:

1. Select an entity that has a boundary condition using the proper selection tool.
2. Select *Gen2DM | Delete BC*.

Once the model interface has been defined, mesh created and assigning boundary conditions assigned to entities has begun, it is recommended that the model definition should remain untouched. The manipulation of the boundary condition definitions (i.e. deleting types or type variables) may interfere with proper simulation set up and cause unforeseen problems.

6.8 Assigning Material Properties

Each element in the mesh is assigned a material type ID. This geometry has two material types. To see each of these materials:

1. Click the *Display Options*  macro or select *Display | Display Options* or press *CTRL+d*.
2. In the *2D Mesh* tab, turn on *Materials*.
3. Turn off *Nodes*, *NodalBC*, *Nodestrings*, and *Elements*.
4. Click the *OK* button to close the *Display Options* dialog.

The mesh boundary and materials should be on showing the sides and the main channel of the river.

Before continuing, turn off the material display. To do this:

- Open the *Display Options* dialog. Turn off *Materials* and turn *Elements* back on.

The materials were created with two attributes. The material properties define how water flows through the element. To edit the material parameters:

1. Select *Gen2DM | Material Properties*.
2. Click the *General Material Properties* button.
- ❖ Note: This dialog can also be entered into by selecting *Edit | Material Data*.
3. Change *Material 01* to be called “Main channel” instead.
4. Change *Material 02* “Side channel”.
5. Click the *OK* button to close the *Material Data* dialog.
6. Highlight *Main channel*.
7. Enter 0.030 and 20.0 for *Manning* and *Kinematic eddy viscosity*, respectively.
8. Highlight *Side channel*.
9. Enter 0.045 and 20.
10. Click *OK*.

The eddy viscosity and roughness parameters have now been defined for this model.

6.9 Gen2DM Model Check

SMS can detect anomalies within the mesh and model definitions, by performing a quick check. Not all invalid situations can be distinguished because Gen2DM is a user defined interface for an executable outside SMS. However, SMS will look for basic mesh problems and missing model definitions. To run this check:

1. Select *Gen2DM | Check Model*.
2. The *Model Checker* dialog should appear and give a list of potential issues. The highlighted *Problem* will include a *Description* and a *Fix*. Read the description and follow the fix instructions if necessary by clicking the *Done* button and performing the required operations.
 - ❖ Note: If no problems are found, a message stating such will be displayed. Click *OK*.
3. After performing any set of procedures, rerun the checker until satisfied.
4. Manually check model variables for validity as suggested by any documents included with the model executable.
5. Save the file as “Conestoga_River_Sim” by selecting *File | Save As*.
6. Change the *Save as type* to *2D Mesh Files (*.2dm)* and enter the name.
7. Click the *Save* button to save the simulation.

6.10 Gen2DM (*.2dm) File Format

SMS uses a generic format that is extremely flexible to write out “*.2dm” file. The file could be opened in a text editor (that does not trim line length) and easily edited with basic knowledge of the *cards* used. The file is organized by *key words at the beginning of each line* followed by data describing attributes. If you opened *Conestoga_River_Sim.2dm* using Microsoft® *Notepad*, you would see the following format (cards shown in proper order with data and description):

- o MESH2D
 - Denotes the beginning of the mesh data – no data
- o E3T 94 1103 1101 1102 2
 - A linear triangular element (3 vertices) – element ID (integer), 1st vertex node ID (integer), 2nd vertex node ID (integer), 3rd vertex node ID (integer), element material type ID (integer)

o E4Q 95 103 388 387 104 2

- A linear quadratic element (4 vertices) – element ID (integer), 1st vertex node ID (integer), 2nd vertex node ID (integer), 3rd vertex node ID (integer), 4th vertex node ID (integer), element material type ID (integer)

o E6T 17 7 1254 25 1255 8 1256 2

- A quadratic triangular element (3 vertices, 3 mid points) – element ID (integer), 1st vertex node ID (integer), 1st mid point node ID (integer), 2nd vertex node ID (integer), 2nd midpoint node ID (integer), 3rd vertex node ID (integer), 3rd mid point node ID (integer), element material type ID (integer)

o E8Q 18 21 1248 7 1256 8 1257 26 1258 2

- A quadratic quadrilateral element (4 vertices, 4 mid points) – element ID (integer), 1st vertex node ID (integer), 1st mid point node ID (integer), 2nd vertex node ID (integer), 2nd midpoint node ID (integer), 3rd vertex node ID (integer), 3rd mid point node ID (integer), 4th vertex node ID (integer), 4th mid point node ID (integer), element material type ID (integer)

o E9Q 50 28 1315 53 1316 52 1317 27 1260 4630 2

- A quadratic quadrilateral element (4 vertices, 4 mid points, an element centered node) – element ID (integer), 1st vertex node ID (integer), 1st mid point node ID (integer), 2nd vertex node ID (integer), 2nd midpoint node ID (integer), 3rd vertex node ID (integer), 3rd mid point node ID (integer), 4th vertex node ID (integer), 4th mid point node ID (integer), the element centered node ID (integer), element material type ID (integer)

❖ Note: The *E3T*, *E4Q*, *E6T*, *E8Q* and *E9Q* cards may be intermixed in the written order. The mesh is written by starting with one element and listing neighboring elements. The node ID for these cards are in counterclockwise order around the element (ending with the centroid node for *EQ9s*). Each card will be written only if an element of the type is present in the mesh.

o ND 3260 -7.62822790e+001 4.00306200e+001 7.40765873e+001

- A node – node ID (integer), x position (real number), y position (real number), z position (real number)

❖ Note: The xyz positions are positive/negative one digit with eight decimal places followed by “e” (times ten to the...) positive/negative power real numbers.

o NS 567 2489 568 2545 594 2626 630 2702 -663

- A nodestring – nodestring head node ID, 1st vertex node ID (if any),... last vertex node ID (if any), nodestring tail node ID

- ❖ Note: The nodestring tail is denoted by a negative sign in front of the tail node ID. A nodestring may consist of more than ten nodes which constitute a file line, consequently a nodestring may extend multiple *NS* cards. Each sequential line should be read until the negative tail node ID is found, ending the nodestring definition. If no nodestrings are present in the mesh, this card will not be written.
- ❖ Note: Node IDs for all cards above are limited to six digits (i.e. 999999 maximum).
- o BEGPARAMDEF
 - Denotes the end of the mesh data (begun by *MESH2D*) and the beginning of the Gen2DM model definition – no data
- o GM "Gen2DM"
 - Gen2DM model name – model name (text)
- ❖ Note: Text is always delimited by quotation marks.
- o SI 0
 - Using International System of Units – 0 (false) or 1 (true)
- o DY 1
 - Dynamic model – 0 (false) or 1 (true)
- o TU "minutes"
 - Time units –time unit name (text)
- o TD 20 1000
 - Dynamic time data – time step length (integer/real number), maximum model time (integer/real number)
- ❖ Note: *TD* data is written as an integer when possible or written as a real number if decimal places are not all zeroes.
- o KEY "sms-gen2dm"
 - Gen2DM model definition informal security password – key (text)
- ❖ Note: This card is only written when the model definition is locked at time of saving.
- o PG "Hydrodynamic" 1
 - Parameter group – group name (text), activity state: 0 (false) or 1 (true)

- ❖ Note: If the parameter group is inactive, then the parameters defined following this card until the next *PG* or *NUME* card is also inactive and a component of this group.
- o PD "Critical scour velocity" 4 "2.0 ft/sec"
 - Parameter definition – parameter name (text), parameter type (integer),...
 - If the parameter type is zero (boolean type): ...default value: 0 (false) or 1 (true)
 - Type is one (integer type): ...default value (integer), minimum value (integer), maximum value (integer)
 - Type is two (real number type): ...default value (real number), minimum value (real number), maximum value (real number)
 - Type is three (text type): ...default value (text)
 - Type is four (option type): ...default selection (text)
- o PO "0.8 ft/sec" "2.0 ft/sec" "2.6 ft/sec"
 - Parameter option definition – 1st option (text), 2nd option (text) (if any), ...last option (text)
- ❖ Note: The *PO* card is only written directly following a *PD* card with a parameter type of four. The default value of the option parameter should be one of the options list in the *PO* card line.
- ❖ Note: Maximum/minimum integer values of 2147483648 and -2147483647 are understood as no bounds. The integer limits are characterized by four bytes. Maximum/minimum real number values of 1.79769e+308 and -1.79769e+308 are understood as no bounds. The real number limits are characterized by eight bytes.
- o NUME 3
 - Number of entities – number of entities available: nodes, nodestrings and elements (integer)
- o BCPGC 1
 - Boundary condition parameter group correlation – whether allowing boundary conditions to be correlated to parameter groups: 0 (false) or 1 (true)
- o BEDISP 0 255 127 0 1 0 1 1 255 127 0 1

- Boundary entity display options – entity type ID (integer), entity label font style (integer), entity label color: red value (integer), entity label color: green value (integer), entity label color: blue value (integer), whether entity label is active: 0 (false) or 1 (true), whether entity label show boundary condition values: 0 (false) or 1 (true), inactive entity size (integer), inactive entity style (integer), inactive entity color: red value (integer), inactive entity color: green value (integer), inactive entity color: blue value (integer), whether inactive entity is active: 0 (false) or 1 (true)
- o BEFONT 1 -19 0 0 0 700 255 0 0 0 3 2 1 49 "Courier New"
- Boundary entity label font – entity type ID (integer), font information (integers and text) (if necessary)
- ❖ Note: The font information may be represented by a single integer or represented by multiple integers and a font name.
- ❖ Note: The *BEDISP* and *BEFONT* cards are written before the entity's boundary condition and values (if any) are written. The next instance of these cards will begin the next entity. The entity type ID equals zero for node, one for nodestring and two for element.
- o BD 1 "Water surface" 1 3 "Elevation" "Essential/Natural factor" "Vary along nodestring factor" 0 "(none)"
- Boundary condition definition – entity type ID (integer), condition type name (text), condition ID (integer), number of values condition contains (integer), 1st value name (text) (if any), 2nd value name (text) (if any),... last value name (text) (if any), condition is legal on the interior of the mesh: 0 (false) or 1 (true), name of parameter group this condition is correlated with (text)
- ❖ Note: If the *BCPGC* card reads false, then the parameter group the condition is correlated with is not read. A group correlation name of *(none)* means the condition is a global condition and not parameter group specific.
- o BV "Elevation" 0 -1.79769e+308 1.79769e+308
- Boundary condition value – value name (text), default value (integer/real number), minimum value (integer/real number), maximum value (integer/real number)
- ❖ Note: The *BV* card follows directly after a *BD* card that has a positive non-zero number of values. A *BV* card for each value name in the *BD* should be written.
- o BCDISP 1 5 1 1 0 0 0 1
- Boundary condition display options – entity type ID (integer), boundary condition ID (integer), , condition entity size (integer), condition entity style (integer), condition entity color: red value (integer), condition entity color:

green value (integer), condition entity color: blue value (integer), whether condition entity is active: 0 (false) or 1 (true)

- ❖ Note: A *BCDISP* card should follow every *BD* card.
- o MD 2 "Manning" "Kinematic eddy viscosity"
 - Material property definition – number of properties (integer), 1st property name (text), 2nd property name (text) (if any),... last property name (text) (if any)
- o MV "Manning" 0.035 0.01 0.18
 - Material property value (attribute) – attribute name (text), default value (integer/real number), minimum value (integer/real number), maximum value (integer/real number)
- ❖ Note: The *MV* card follows directly after a *MD* card that has a positive non-zero number of values. A *MV* card for each value name in the *MD* should be written.
- ❖ Note: The *BD*, *BV*, *BCDISP*, *MD* and *MV* cards are only written if data exists for each.
- o ENDPARAMDEF
 - Denotes the end of the Gen2DM model definition (begun by *BEGPARAMDEF*) – no data
- o BEG2DMBC
 - Denotes the beginning of the Gen2DM model assignment – no data
- o MAT 1 0.03 20
 - Material assignment – element material type ID (integer), 1st material property attribute value (integer/real number) (if any), 2nd attribute value (integer/real number),... last attribute value (integer/real number) (if any)
- o GG "Hydrodynamic"
 - Parameter group name – group name (text)
- o GP "Time interval" 150
 - Parameter assignment – parameter name (text),...
 - If the parameter is defined as type boolean: ...parameter value: 0 (false) or 1 (true)
 - Defined as type integer: ...parameter value (integer)

- Defined as type real number: ...parameter value (integer/real number)
 - Defined as type text: ...parameter value (text)
 - Defined as type option: ...parameter value selected (text)
- ❖ Note: The *GP* card directly follows the *GP* card the parameter are contained in.
- o BCN 863 3
 - Nodal boundary condition assignment – node ID (integer), boundary condition type ID (integer), 1st condition value (integer/real number) (if any), 2nd condition value (integer/real number) (if any),... last condition value (integer/real number) (if any)
 - o BCS 3 4 1 1 0 0
 - Nodestring boundary condition assignment – nodestring ID (integer), boundary condition type ID (integer), 1st condition value (integer/real number) (if any), 2nd condition value (integer/real number) (if any),... last condition value (integer/real number) (if any)
 - o BCE 1235 1 1 0 0
 - Element boundary condition assignment – element ID (integer), boundary condition type ID (integer), 1st condition value (integer/real number) (if any), 2nd condition value (integer/real number) (if any),... last condition value (integer/real number) (if any)
 - o TIME 20
 - Dynamic model time stamp – current time step (integer/real number)
- ❖ Note: This card only is written if the model is dynamic and a boundary condition is defined dynamically using a curve. All entity boundary condition assignment cards (*BCNs*, *BCSs*, *BCEs*) following this card until the next instance of the *TIME* card is assigned for this time step.
- ❖ Note: The *MAT*, *GG*, *GP*, *BCN*, *BCS*, *BCE* and *TIME* cards are only written if data exists for each.
- o END2DMBC
 - Denotes the ending of the Gen2DM model assignment (begun by *BEG2DMBC*) – no data

This concludes the “*.2dm” file format.

6.11 Exporting the Gen2DM File and Running the Model

Exporting the Gen2DM file consists of simply saving the file. All definitions and assignments are contained in it. Use your computer operating system to move or copy the “*.2dm” file as necessary and run the model executable. Follow the instructions for the executable concerning location of the file and how to import to compute the model simulation. If a desired executable outside of SMS will not recognize the Gen2DM file format, contact the model distributor or designer for information on new versions or an additional executable to reformat the “*.2dm” file into a compliant form.

6.12 Conclusion

This concludes the Generic 2D Mesh Model (Gen2DM) tutorial. You may continue to experiment with the SMS interface or you may quit the program. To quit SMS at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

Feature Stamping

7.1 Introduction

In this lesson you will learn how to use conceptual modeling techniques to create numerical models that incorporate flow control structures into existing bathymetry. The flow control structures you will be creating are abutments for a proposed bridge over Double Pipe Creek near Detour, Maryland. To do this you will be using feature stamping.

7.2 Opening a Background Image

To provide a base map and to help you place the centerlines for the abutments of the proposed bridge you will open an aerial photograph of Double Pipe Creek near Detour, Maryland. To open the image:

1. Select *File | Open*.
2. Select *DoublePipeCreekPhoto.jpg* in the *tutorial\Feature Stamping* directory and click the *Open* button.
3. Depending on your preference settings, *SMS* may ask if image pyramids are desired. It is advised that you select the toggle to not ask this question again and click *Yes*.

SMS displays the aerial photograph (Figure 7-1).

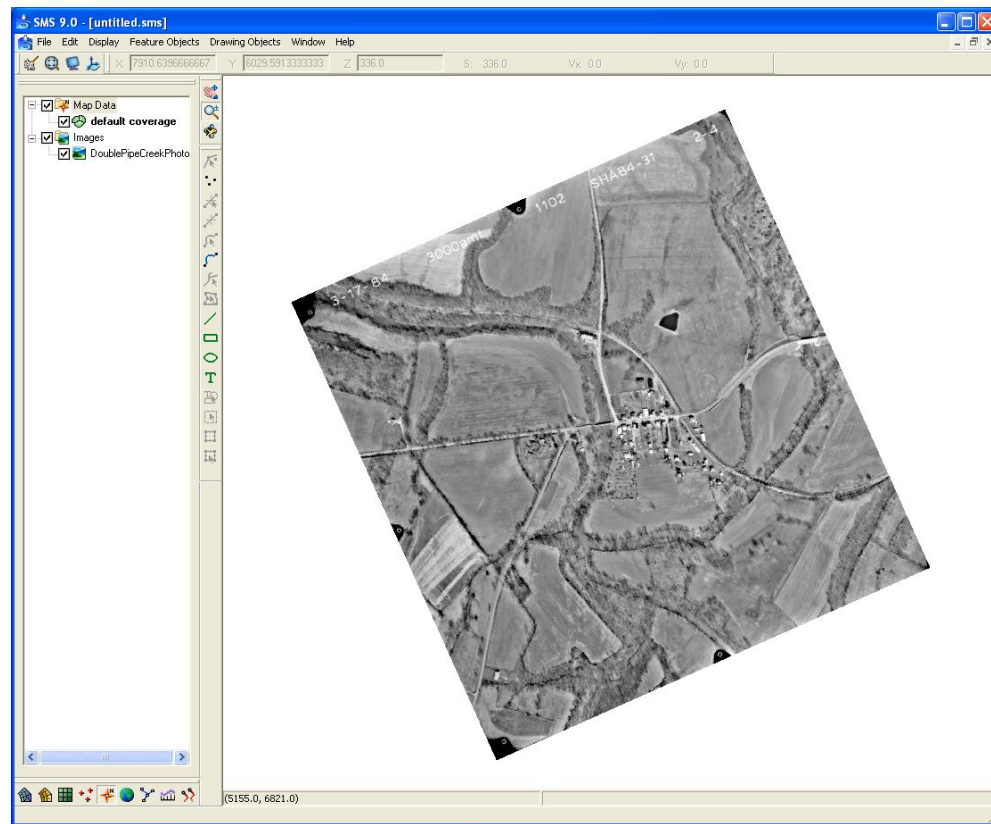


Figure 7-1 Aerial photograph of Double Pipe Creek near Detour, Maryland

7.3 Specifying the Coordinate System

The image has now been read into *SMS*, but *SMS* has not been told what coordinate system the data is referenced to. The coordinate system is dependent on the data source. To specify the coordinate system:

1. Select *Edit | Current Coordinates*.
2. Leave the *Horizontal System* as “Local”, but change the *Horizontal* and *Vertical Units* to “U.S. Survey Feet.”

7.4 Importing Bathymetric Data

For this lesson you will use bathymetry from a survey of the area around Double Pipe Creek near Detour, Maryland before construction of the elevated road and bridge. To bring the survey data into *SMS*:

1. Select *File | Open*.

2. Select *detour.xyz* and click the *Open* button.
3. The *File Import Wizard* dialog will appear. Click *Next* to proceed to step 2 of the *File Import Wizard*.
4. Click *Finish* to close the *File Import Wizard* and import the survey data.
5. Next go into the Display Options Menu. In the Scatter section switch off the points and select contours. Click on the Contour Options tap and switch the color method to Color Fill and change the transparency to .5 and click OK.

This survey file contains elevation data for Double Pipe Creek and its floodplain which includes the town of Detour, Maryland. The survey data has already been adjusted to the same local coordinate system as the image. Transparent contours of the survey points displayed over the background image are shown in Figure 7-2.

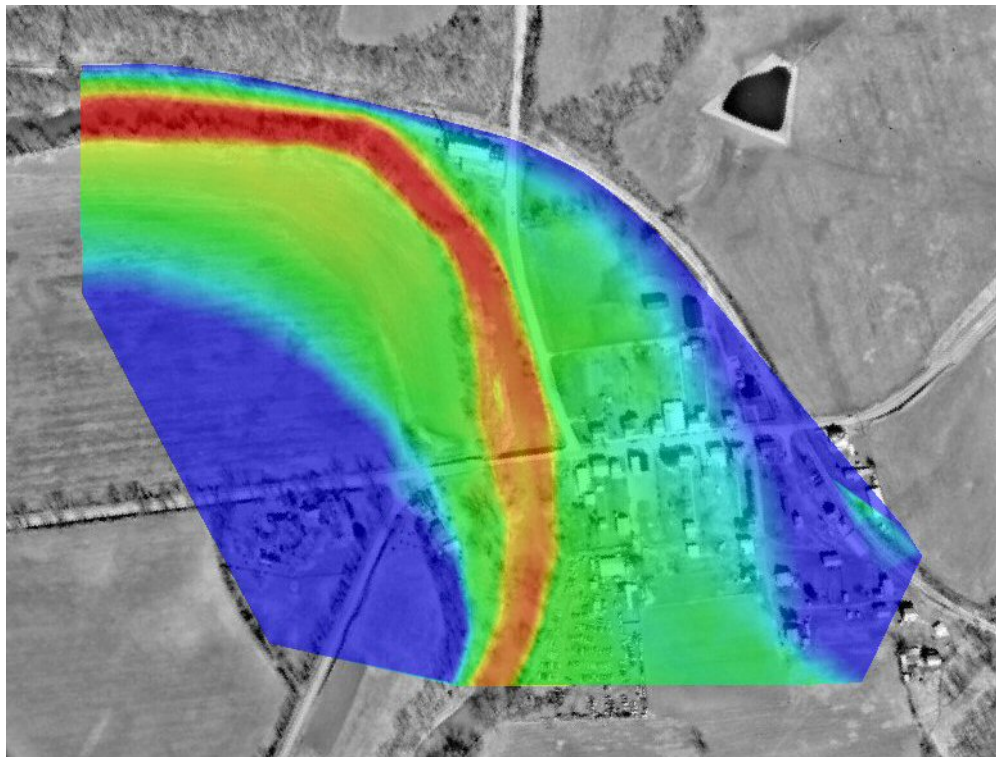


Figure 7-2 Bathymetry for Double Pipe Creek and its floodplain

7.5 Creating the Model Domain

Before creating a numerical model, a conceptual model will be created to define the extents of the model domain. By using a conceptual model, you can take advantage of automatic meshing algorithms. The two sides of the model domain running along the length of Double Pipe Creek will be formed by extracting the 330 foot contour from the survey data. The ends of these two boundaries will then be connected to

create the upstream and downstream boundaries of the model domain. To define the model domain:

1. Right-click on the *default coverage* coverage  in the *Project Explorer* and select the *Rename* menu item. Rename the coverage “Double Pipe Bridge.”
2. Right-click on the *Double Pipe Bridge* coverage  in the *Project Explorer* and select the *Type | TABS* menu item to specify that this coverage will be used to develop a conceptual model for the *TABS* package.
3. Right-click on the *detour* scatter set  in the *Project Explorer* and select the *Convert | Scatter Contours -> Map* menu item.
4. Enter an *Elevation* of 330 feet and a *Spacing* of 100 feet in the *Create Contour Arcs* dialog.
5. Click *OK* to close the *Create Contour Arcs* dialog and generate arcs along the 330 foot contour. The resulting arcs run along the length of Double Pipe Creek. A single looped arc is created on the extreme east side of the scatter set. Delete this arc to do this you will need to switch to map module if it isn't already.
6. With the *Create Feature Arc*  tool create the upstream and downstream boundaries of the model domain as shown in Figure 7-3. Delete any dangling arcs that result when creating these two boundaries. Note the arcs are not to be placed on the ends of the existing arcs.

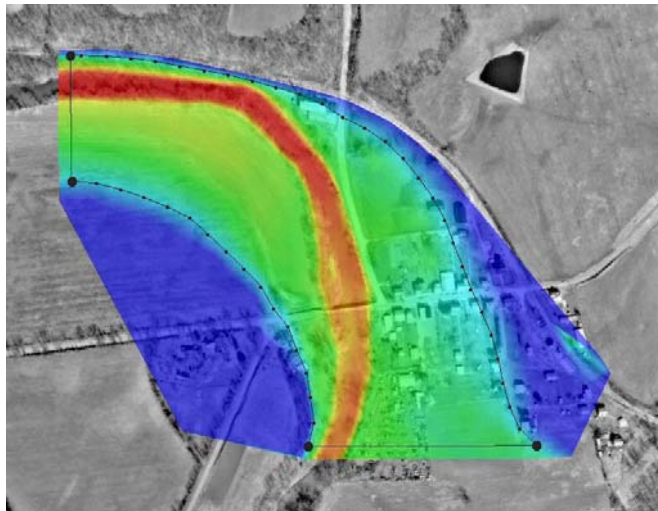


Figure 7-3 Model domain of Double Pipe Creek(I resized this picture so it could still fit on this page.)

The model domain extents are now defined in the *Double Pipe Bridge* coverage. It is important to note that when creating a finite element mesh from a conceptual model, the bathymetry is interpolated from the scatter set. Therefore, the conceptual model should be within the bounds of the scatter set to avoid difficulties that arise when extrapolating data.

7.6 Creating the Abutments

As mentioned above the abutments of the proposed roadway will be created using feature stamping. This lesson presents stamping the abutments for the proposed bridge over Double Pipe Creek in five steps: setting up a *Stamping* coverage, positioning the abutments, specifying the geometry of the abutments, stamping the abutments into the existing bathymetry and adding the stamped features to the conceptual model.

7.6.1 Setting Up a Stamping Coverage

SMS includes a coverage type, called a *Stamping* coverage, for positioning and defining the geometry of features to be forced into existing bathymetry using feature stamping. To setup the *Stamping* coverage for this lesson:

1. Create a new coverage, in map data, name it “Feature Stamp”, set its *Type* to *Stamping* and activate it.
2. Right-click on the *Feature Stamp* coverage  in the *Project Explorer* and select the *Properties* menu item to bring up the *Stamping Coverage Attributes* dialog.
3. In the *Stamping Coverage Attributes* dialog select the *elevation (Z)*  item in the tree control and leave the *Bathymetry Type* as “Elevation.” This sets the *elevation (Z)* dataset of the *detour* scatter set as the bathymetry the stamped features will be forced into. Furthermore, by leaving the *Bathymetry Type* as “Elevation,” you tell SMS that the selected dataset contains elevation values rather than depth values.
4. Click OK to close the *Stamping Coverage Attributes* dialog.

7.6.2 Positioning the Abutments

You will position the abutments by creating feature arcs along their centerlines. The accuracy in how the abutments intersect the existing bathymetry depends on how many vertices are distributed along the centerline arcs. For this lesson you will distribute the vertices so they are closer together where the slope of the bathymetry changes rapidly near the banks of the creek and further apart where the slope is nearly flat in the floodplain. To create the centerline arcs for the abutments:

1. Using the *Create Feature Arc*  tool create arcs representing the centerlines of the two abutments as shown in Figure 7-4. You can use the roadway in the aerial photograph to help you position the centerline arcs and line them up with each other. Create the arcs starting outside the model domain in the floodplain and proceeding toward the Double Pipe Creek. End the arcs at the edge of the creek. The length of the bridge will be roughly the distance

between the two nodes. You may have to adjust your display options  or zoom  in to better see the model domain on the *Double Pipe Bridge* coverage.

2. With the *Select Feature Arc*  tool, select the centerline arc for the west abutment and select *Feature Objects* | *Redistribute Vertices*. This brings up the *Redistribute Vertices* dialog.
3. In the *Redistribute Vertices* dialog set *Specify* to “Number of Segments,” the *Num Seg* to 6 and the *Bias* to 0.1. Click *OK* to close the *Redistribute Vertices* dialog.
4. Redistribute the vertices along the centerline for the east abutment in a similar manner using 10 segments.

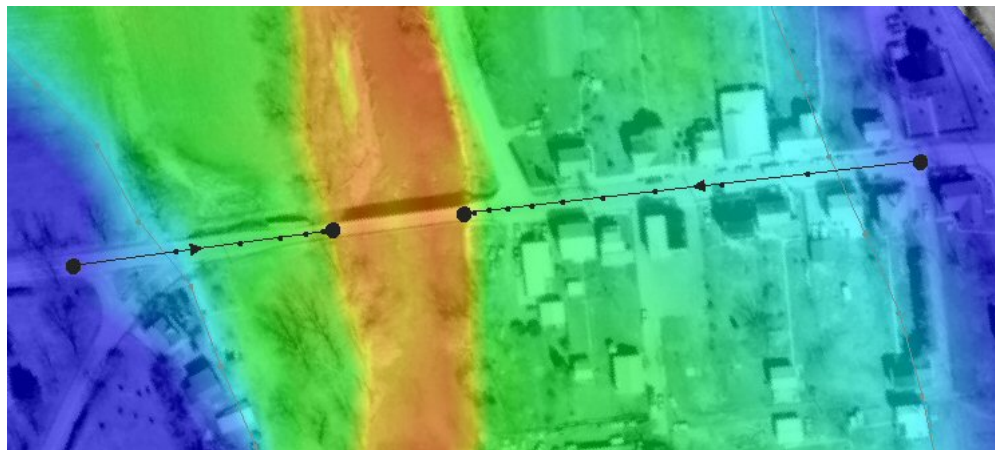


Figure 7-4 Abutment centelines for the proposed bridge over Double Pipe Creek

7.6.3 Specifying the Geometry of the Abutments

Now that the abutments have been positioned with centerline arcs you can specify their geometry. To specify the geometry of the abutments:

1. With the *Select Feature Arc*  tool double-click on the west abutment. This brings up the *Stamping Arc Attributes* dialog.
2. In the *Stamping Arcs Attributes* dialog specify the *Feature Name* as “West Abutment.” Leave the *Stamping Type* as “Fill Feature” since this abutment will be increasing the elevation of the existing bathymetry.
3. Click the *Constant -> Elevation* button in the *Centerline (CL) Profile* area to bring up the *Constant -> Elevation* dialog. Enter a constant elevation of 332 feet and click *OK* to close the dialog. This sets the elevation at each of the points along the centerline arc to 332 feet. The elevations for the points along

the centerline can be set one at a time in the *Centerline (CL) Profile Spreadsheet* or all at once using the macros found below this spreadsheet.

Note that the first point along the centerline is marked with an arrow ►. This arrow identifies the current point. The *Cross-sections (CS)* area of the attributes dialog displays the cross-section for the current point for viewing and editing. When the current point is changed by clicking on it in the *Centerline (CL) Profile Spreadsheet*, the *Cross-sections (CS)* area updates to display the cross-section of the new current point. You will now specify the cross-sections at each point along the centerline.

4. In the *Cross-sections (CS)* area click the *Specify Top Width and Single Side Slopes* macro button to bring up the *Top Width and Side Slopes* dialog. Enter a *Top Width* of 25 feet and *Left* and *Right Slopes* of -1. Click *OK* to close the dialog. A simple cross-section has now been specified. To ensure these cross-sections intersect the bathymetry when being stamped, specify a *Maximum Distance from CL* of 35 feet for both the left and right sides of the cross-section.
5. Copy this cross-section to the remaining centerline points by clicking the *Current CS -> All CS* macro button. Click *Yes* when prompted to adjust the cross-sections based on the centerline elevation.
6. To specify a slope on the end of the abutment click the *Last End Cap* button to bring up the *Last End Cap* dialog. Leave the *Type of end cap* as “Sloped Abutment” and the *Angle* as 0.0°. In the last line of the *Slope Spreadsheet* enter a *Distance from CS* of 1 foot and an *Elevation* of 331 feet. To ensure the sloped abutment intersects the bathymetry, specify a *Maximum Distance from CS* of 25 feet.
7. Click *OK* twice to exit the *Last End Cap* and *Stamping Arc Attributes* dialogs.
8. Repeat steps 1 through 7 for the east abutment except set the *Feature Name* to “East Abutment.”

The geometry for both the abutments has now been specified. For this lesson you are creating fairly simple features to force into the existing bathymetry. The feature stamping interface inside SMS has been designed to make creating these simple features quick but at the same time allowing for the creation of more complex features. You are now ready to stamp the abutments and add them to the conceptual model.

7.6.4 Stamping the Abutments

To maintain the integrity of the conceptual model and the existing bathymetry, feature stamping creates a new coverage and a new scatter set for each stamped feature. If the conceptual model or the existing bathymetry becomes corrupted, it

makes it difficult to test several scenarios for the placement and geometric design of features being stamped. To stamp the abutments into the existing bathymetry:

1. Select *Feature Objects* | *Stamp Features* to bring up the *Stamp Features* dialog.
2. In the *Stamp Features* dialog make sure the *Coverage Type* is set to “TABS” and click *Stamp*.

You will notice a coverage and scatter set are added to the *Project Explorer* for the west abutment and also for the east abutment.

7.6.5 Adding the Abutments to the Conceptual Model

A stamped feature can be incorporated into the conceptual model in a number of ways. This includes either using the scatter set of the stamped feature, the coverage of the stamped feature or both. These three methods are described in the following paragraphs.

To use just the scatter set of the stamped feature as part of the conceptual model, it must be merged with the scatter set of the existing bathymetry. The merged scatter set is then identified as the source of elevation data for your numerical model. Since the element edges of a mesh will not necessarily match the intricate slope changes of the feature, this method is not recommended for models requiring detailed results around the stamped features. Such detailed analysis requires greater control over how the feature and the area around the feature are meshed.

Figure 7-5 shows a rotated view of the west and east abutments on top of the existing bathymetry. This is done by using the rotate  and zoom  tools.

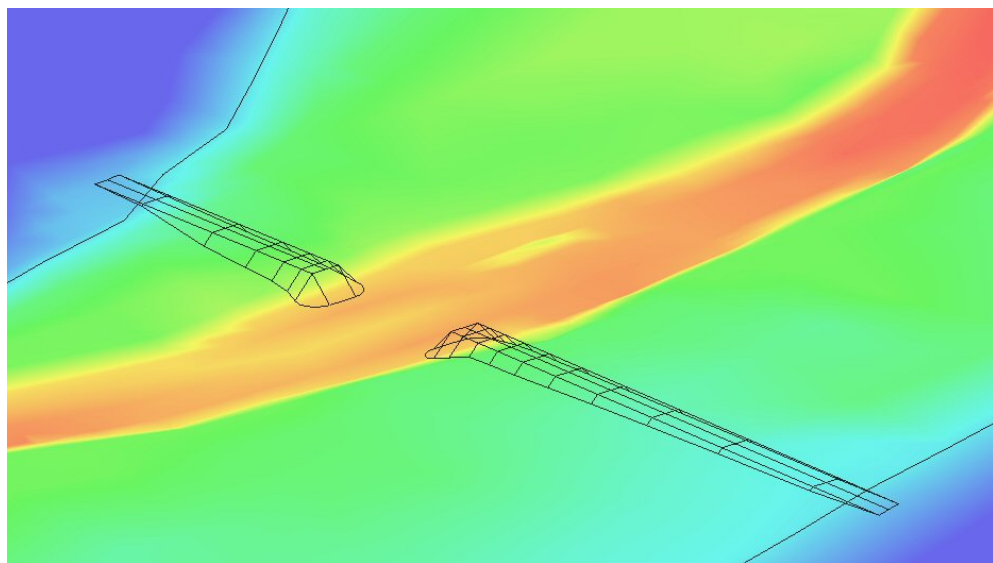


Figure 7-5 The east and west abutments stamped into the existing bathymetry

7.7 Conclusion

This concludes the Feature Stamping tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

Mesh Editing

This tutorial lesson teaches manual finite element mesh generation techniques that can be performed using *SMS*. It gives a brief introduction to tools in *SMS* that are useful for editing a finite element mesh. The mesh in this tutorial will be created by hand from survey points. These mesh editing methods should be used in conjunction with map module meshing to generate a good finite element mesh. This tutorial exists to show useful tools for editing small portions of a mesh after the mesh generation. All files for this tutorial are in the *tutorial\tut07* directory.

8.1 Importing Topographic Data

Data points for a finite element mesh can be generated directly from topographic data, such as a list of survey points. An *XYZ* file contains the header *XYZ* on the first line of the file and then the *X*, *Y*, and *Z* coordinates of each point on a single line in the file. This type of file can be opened by *SMS*. To open the *poway1.xyz* file:

1. Select *File | Open*.
2. Change to the *tutorial\tut07* directory and open the file *poway1.xyz*.
3. The *File Import Wizard* will come up. Click *Next* in *Step 1*.
4. In *Step 2* change the *SMS data type* to *Mesh*, and uncheck the *Triangulate data* option.
5. Make sure the column headings for the three data columns show a mapping of (ie. The *Type*) are *X*, *Y* and *Z* respectively.

6. Click *Finish* to import the data points.

The data points from the file are converted to *mesh nodes*. From the *File Import Wizard*, a user can open any columnar data into SMS. See the *SMS online Help* for more information on the *File Import Wizard*. The data points created from *poway1.xyz* are shown in [Figure 8-1](#).

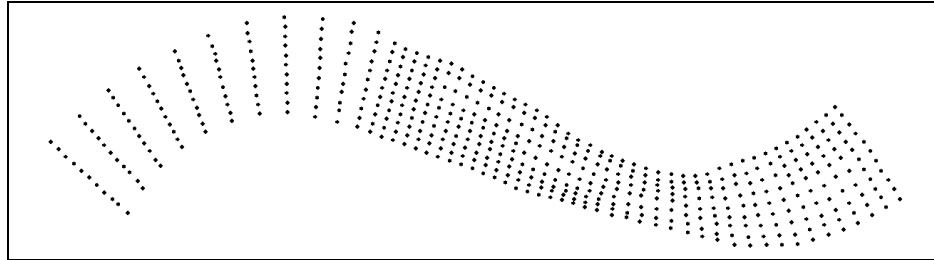


Figure 8-1 The *poway1.xyz* data points.

8.2 Triangulating the Nodes

After nodes have been created, elements are required to build a finite element mesh. Elements connect the nodes to define the extents of the flow area. SMS provides numerous automatic mesh generation techniques. This section will review a very simple technique, *triangulation*. If the *Triangulate data* option had not been unchecked above, this step would have been done automatically when the file was imported. The file would then have looked like [Figure 8-2](#) when it was opened. To create a triangulated mesh from the data points:

1. Select *Elements | Triangulate*.
2. Since no nodes are selected, you will be prompted to triangulate all of them. Click the *Yes* button at this prompt.

When SMS triangulates data points, it creates either quadratic triangles or linear triangles from the mesh nodes. Different numerical models support different types of elements. *RMA2*, *FESWMS* and *RMA10* support quadratic elements, while *HIVEL*, *ADCIRC* and *CGWAVE* support only linear elements. After the nodes are triangulated, the mesh will look like that in [Figure 8-2](#). It may or may not have midside nodes, depending on whether the elements are linear or quadratic.

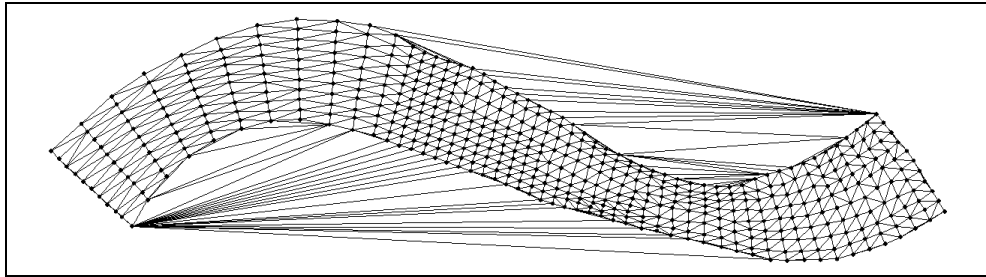


Figure 8-2. The results of triangulating the poway1.xyz data.

8.3 Deleting Outer Elements

The triangulation process always creates elements outside the real mesh boundaries. For this tutorial, the mesh should be in the shape of a rotated S, so any elements outside of this boundary must be deleted. To remove these elements:

1. Choose the *Select Element* tool  from the *Toolbox*.
2. Click on an element to select it.
3. Select another element by holding the *SHIFT* key and clicking on it.
4. Select *Edit | Delete* or press the *DELETE* key to remove the selected elements.
5. Refresh the display.

It is tedious to individually select every element that needs to be deleted. *SMS* provides a hot key to help selecting groups of adjacent elements. To select a group of adjacent elements:

1. Choose the *Select Element* tool  from the *Toolbox*.
2. Hold the *CTRL* key.
3. Click and drag a line through some elements to select them. Be careful to only select elements outside the S shape.
4. Select *Edit | Delete* or press the *DELETE* key to remove the selected elements.
5. Refresh the display.

Continue deleting elements that are outside the boundaries of the S shape.

8.4 Deleting Thin Triangles

It is not uncommon for the triangulation process to create very thin triangular elements outside the desired mesh boundary. The three corner nodes of thin triangles are almost collinear and the elements may be too thin to see or select. If these are not deleted, numerical errors in the model solution can result.

SMS provides a way to define what is meant by a thin triangle using the element *aspect ratio*. The element aspect ratio is the ratio of the element width to its height. Perfect equilateral triangles have an aspect ratio of 1.0 while that of thin triangles is much less. To define the element aspect ratio:

1. Select *Elements | Options*.
2. Set the aspect ratio in the *Select thin triangle aspect ratio* box to 0.1. (The default value is 0.04.) Triangular elements with an aspect ratio less than this are considered to be thin triangles.
3. Click the *OK* button.

The best aspect ratio to use for selecting thin triangles depends on the finite element mesh. For this mesh, the distribution of nodes is rather uniform, so a large aspect ratio will suffice. After this value is set, SMS can check for and select thin triangles. To delete any remaining thin triangles:

1. Select *Elements | Select Thin Triangles*. The lower right portion of the *Status Bar* in the *Graphics Window* shows how many elements became selected due to this operation, along with the total area of the selected elements. There may be quite a few elements selected.
2. Select *Edit | Delete* or press the *DELETE* key. If prompted to confirm, click the *Yes* button.
3. Refresh the display.

The mesh should now look like [Figure 8-3](#).

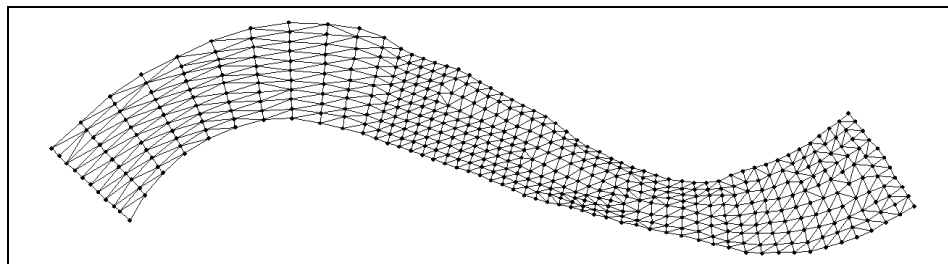


Figure 8-3. The poway1 mesh after deleting excess triangles.

8.5 Merging Triangles

The mesh is composed entirely of triangles. Both ADCIRC and CGWAVE support only triangles. If you are using one of these models, you may skip this section of this tutorial.

Using quadrilateral elements can reduce the number of elements required for a simulation and speed up analysis when using *RMA2*, *RMA10*, *FESWMS* or *HIVEL* because:

- A quadrilateral element covers more area than a triangular element.
- A quadrilateral can maintain good interior angles (90 degrees) and still have high resolution in one direction. This makes these elements more numerically stable.

SMS can automatically merge a pair of triangles into a quadrilateral. Before merging triangles, the *Merge triangles feature angle* should be set. To do this:

1. Select *Elements | Options*.
2. Enter a value of 55.0 in the *Merge triangles feature angle* box. (The default value is 65.0.) Two triangles may be merged if all angles of the resulting quadrilateral are greater than the value specified.
3. Click the *OK* button.

The finite element method is more stable and accurate when quadrilateral elements are rectangular and triangular elements are equilateral. Although it is not practical for a mesh to exist entirely of these perfect shapes, the elements should approach these shapes as close as possible. For this reason, *SMS* merges triangles in an iterative manner. First, it merges elements using the angle criterion of 90°. Then, the angle criterion is decreased by a number of steps to the feature angle specified. Slowly decreasing the feature angle and testing all triangles against this specified angle will form the best-shaped elements.

SMS can merge the triangles in either a selected portion of elements or all elements. In order to merge triangles in the entire mesh, no elements should be selected. To merge triangular elements into quadrilateral elements:

1. Select *Elements | Merge Triangles*.
2. Since no elements are selected, you will be prompted to merge all triangles. Click the *Yes* button at this prompt.

With most meshes, as is the case for this example, not all triangles will be merged. The mesh will appear as in [Figure 8-4](#) after *SMS* merges the triangles.

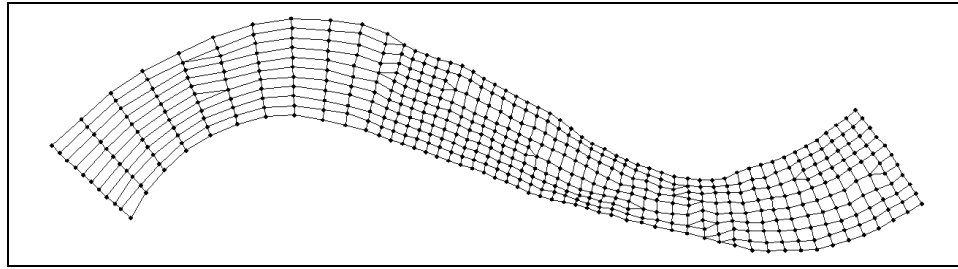


Figure 8-4. The poway1 mesh after merging triangles.

8.6 Editing Individual Elements

After triangulating the nodes, deleting elements outside the boundaries and merging triangles, the mesh often needs further manipulation to add model stability. For a main river channel such as this model, lines of elements should run parallel to the mesh boundary. This is especially important in cases where a portion of the mesh may become dry so that the mesh will dry parallel to the boundary. Two of the tools in SMS used for manipulating individual elements are the *Split/Merge*  tool and *Swap Edge*  tool. With the *Split/Merge* tool, two adjacent triangular elements can be merged into a quadrilateral element or a single quadrilateral element can be split into two triangular elements. With the *Swap Edge* tool, the common edge of two adjacent triangular elements can be swapped. See the *SMS Help* for a better description of these tools.

8.6.1 Using the Split/Merge Tool

Most triangular elements in this mesh were merged into quadrilateral elements when the *Merge Triangles* command was performed in section 8.5. Some of the elements that were not automatically merged can be merged manually. To do this:

1. Zoom into the portion of the mesh shown in [Figure 8-5a](#). Notice the two triangular elements separated by a number of quadrilateral elements.
2. Select the *Split/Merge* tool  from the *Toolbox*.
3. Split the quadrilateral, highlighted in [Figure 8-5a](#), by clicking inside it. There should now be three triangles, as shown in [Figure 8-5b](#).
4. Merge the top two triangles, highlighted in [Figure 8-5c](#), by clicking on the edge between them. (The *Split/Merge* tool should still be selected.)

The result of this split/merge operation is shown in [Figure 8-5d](#). There is now one fewer quadrilateral between the two triangles.

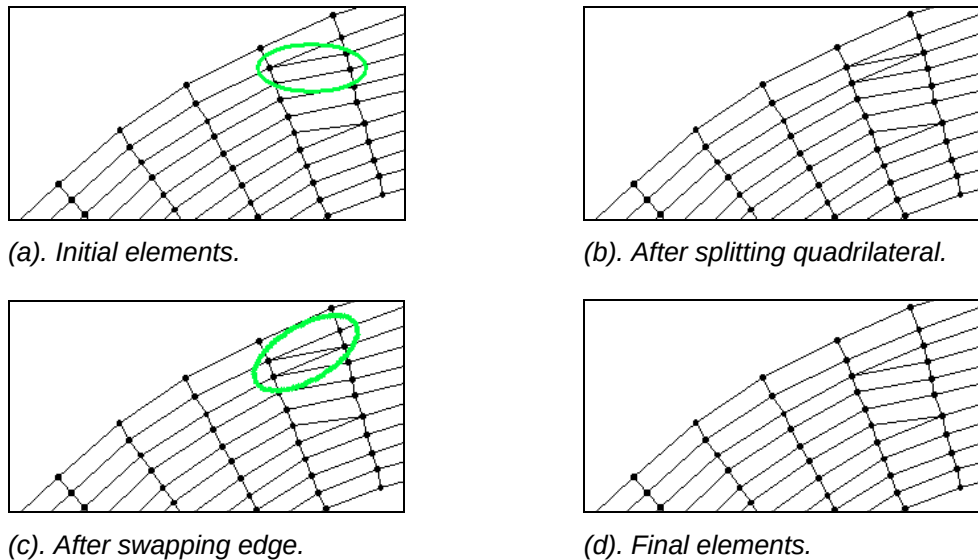


Figure 8-5. Example of manual split / merge procedure.

To finish editing this section:

- Repeat the above split/merge process until there are no more triangles across the section. This part of the mesh should look like [Figure 8-6](#).

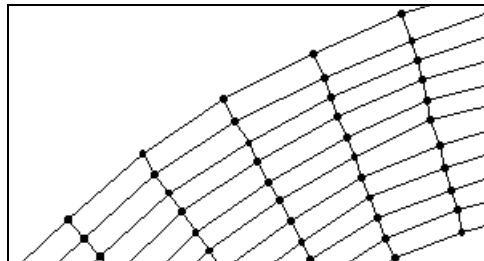


Figure 8-6. The mesh section after merging triangles.

8.6.2 Using the Swap Edge Tool

The common edge between two triangles can be swapped. The best way to understand this is to think of the two triangles as a quadrilateral, and the common edge between them is a diagonal of the quadrilateral. By swapping this common edge, it changes to be along the opposite diagonal of the quadrilateral. If this edge is clicked again, it returns back to its original state. This can be seen in [Figure 8-7](#).

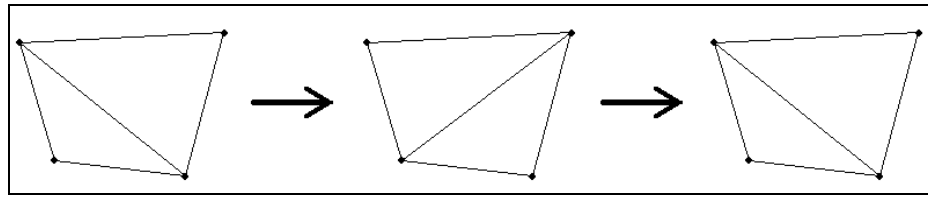


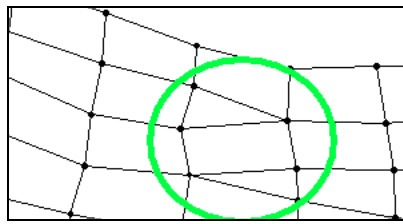
Figure 8-7 The Swap Edge technique.

One place in this mesh requires the use of the *Swap Edge* tool together with the *Split/Merge* tool to be able to merge the triangles. This is located toward the middle of the finite element mesh, at the constriction. The easiest way to find this location is to set the window boundaries to the correct location. To do this:

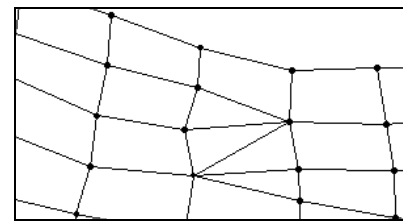
1. Select *Display | View | Window Bounds*.
2. In the *Set Window Boundaries* dialog, select to use the *X range to be specified* option (this will disable the *Y at top* field).
3. Enter these values: *X at left* = 25,200; *X at right* = 25,500; *Y at bottom* = 9,300.
4. Press the *OK* button.

You should now be able to see the portion of the finite element mesh shown in [Figure 8-8](#). In this part of the mesh, there are two triangles that need to be merged together, separated by a single quadrilateral. To do this:

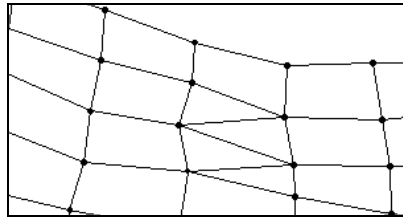
1. Choose the *Split/Merge*  tool from the *Toolbox*.
2. Click inside the quadrilateral, highlighted in [Figure 8-8a](#), that separates the two triangles. The quadrilateral gets split as shown in [Figure 8-8b](#). The new edge was not created in the direction necessary to merge the outer triangles.
3. Choose the *Swap Edge*  tool from the *Toolbox*.
4. Click only once, directly on the edge that was just created inside the quadrilateral. The edge will swap to the other diagonal of the quadrilateral. This result is shown in [Figure 8-8c](#).
5. Once again choose the *Split/Merge*  tool from the *Toolbox*.
6. Merge the top two triangles to form one quadrilateral, and then merge the bottom two triangles to form another quadrilateral. The result is shown in [Figure 8-8d](#).



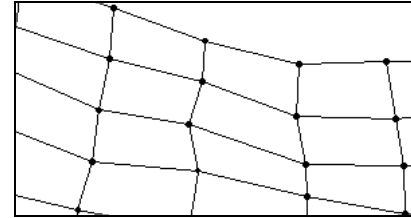
(a). The original elements.



(b). Elements after splitting quad.



(c). Elements after swapping edge.



(d). Final quadrilateral elements.

Figure 8-8 Example of manual swapping procedure.

Although this operation appears simple, it is one that takes some time to get used to performing. Most people do not get through this without making a mistake. However, after you understand this operation, it is easier to use. The *Split/Merge* and *Swap Edge* tools are very useful for manually adjusting small areas of the finite element mesh.

Continue to merge triangles in the areas that you are able to do so. Not all of the triangles can be merged. When you are done, there should be only six triangles left in the finite element mesh, and it should look like that shown in [Figure 8-9](#).

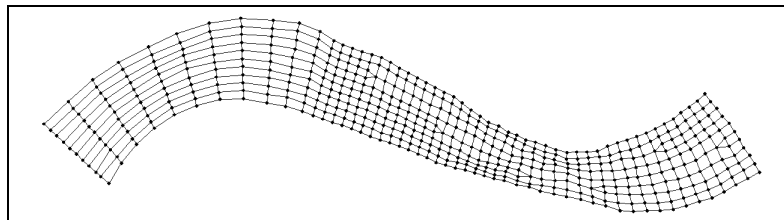


Figure 8-9 The finite element mesh after merging triangles.

8.7 Smoothing the Boundary

When dealing with quadratic finite element meshes, mass loss can occur through a jagged boundary. It is good to smooth the boundary of a quadratic mesh to prevent these losses. Smoothing can only be performed with quadratic models, because the midside nodes get moved while corner nodes do not. SMS currently supports three

quadratic finite element models, *RMA2*, *RMA10* and *FLO2DH*. If you are not using one of these quadratic models, you can skip this section. The quadratic models still support the creation of linear elements. To make sure you have quadratic elements:

1. Select *File* | *Get Info*.
2. In the top right corner of the *Mesh Info* tab, look at the *Element type*. This will be either *quadratic* or *linear*.
3. Click *OK* to close the dialog.
4. If the element type was *linear*, select *Elements* | *Linear <-> Quadratic* to switch the element type. If it was *quadratic*, you are already set.

The easiest way to smooth the entire mesh boundary is by creating a nodestring around the entire mesh boundary. To do this:

1. Choose the *Create Nodestring*  tool from the *Toolbox*.
2. Click the node labeled *Node 1* in [Figure 8-10](#).
3. Hold the *CTRL* key and double-click the node labeled *Node 2* in [Figure 8-10](#). When holding the *CTRL* key, SMS creates a nodestring counter-clockwise around the mesh boundary from the first node to the second node.

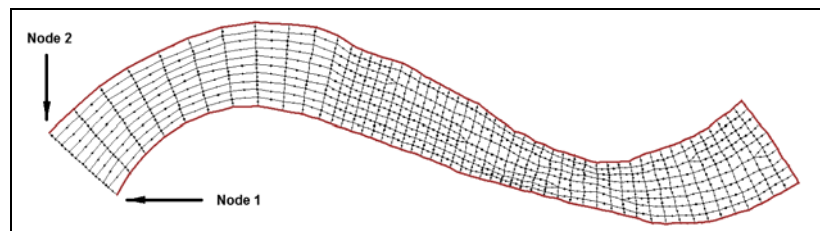


Figure 8-10 The nodestring to create for smoothing.

This nodestring starts from *Node 1*, and runs counter clockwise around the entire boundary to *Node 2*. Notice that this nodestring goes around two sharp corners on the right side of the mesh. To assure that these corners remain sharp:

1. Select *Elements* | *Options*.
2. In the *Element Options* dialog, change the *Smooth nodestring feature angle* to be 45.0. A midside node will not move if it is at a corner that is sharper than this angle.
3. Click the *OK* button.

Now that the nodestring is created and the feature angle is set, the boundary is ready to be smoothed. To do this:

1. Choose the *Select Nodestring*  tool from the *Toolbox*. A small icon will appear at the center of the nodestring (to the right side of the mesh).
2. Click on the icon to select the nodestring. The icon will be filled and the nodestring will be highlighted in red.
3. Select *Nodestrings | Smooth*. The mesh boundary will be smoothed as shown in [Figure 8-11](#).
4. Refresh the display.

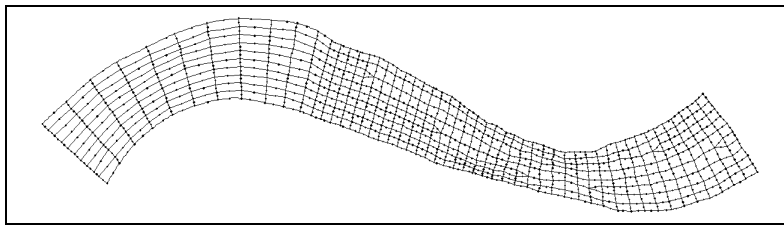


Figure 8-11. Example nodestrings for smoothing the mesh boundary.

In general, it is sufficient to smooth the finite element mesh boundary. However, it may be desirable to further smooth interior elements at sharp bends or where dry elements may change the boundary. Any nodestring can be used for smoothing. See the *SMS Help* for more information on creating interior nodestrings and the smoothing operation.

8.8 Renumbering the Mesh

The process of creating and editing a finite element mesh, as performed in the previous few sections, causes the node and element ordering to become disorganized. This random mesh ordering increases the size of the matrices required by the finite element analysis codes. Renumbering the mesh restores a good mesh ordering, making it faster to run the analysis. Renumbering starts from a nodestring. To renumber this mesh:

1. Choose the *Create Nodestring* tool  from the *Toolbox*.
2. Create a nodestring across the left section, as shown in [Figure 8-12](#).
3. Choose the *Select Nodestring* tool  from the *Toolbox* and select the nodestring that was just created.
4. Choose *Nodestrings | Renumber*. It will not be evident that anything has happened, but the nodes have been numbered from the left to the right. This makes the solution process more efficient and should always be done before running a model.

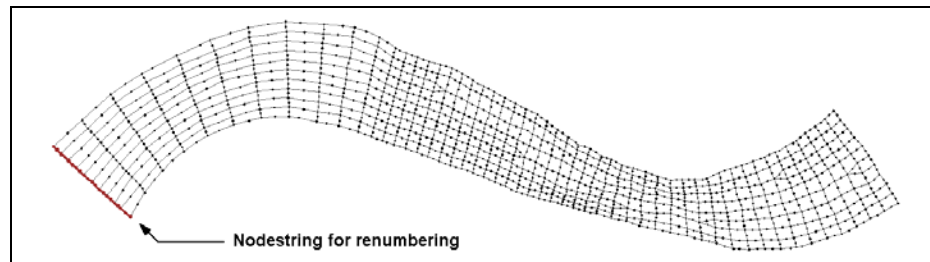


Figure 8-12 The position of the nodestring for renumbering.

When SMS is finished renumbering the mesh, the display will refresh. Remember that adding and deleting nodes or elements changes the mesh order. It is important that renumbering be the last step of the mesh creation process. Editing a mesh invalidates any boundary condition and/or solution files that have previously been saved. (Boundary condition and solution files are discussed in later tutorials.)

8.9 Changing the Contour Options

When the finite element mesh is created, contour lines are drawn to connect points of equal elevation. By default, these contours are displayed as constant green lines. The contour display can be changed using the *Contour Options* dialog. It is always a good idea to look at a color contour map after a new finite element mesh has been created. This helps you better visualize the bathymetry of the model. To set the color fill contours:

1. Choose *Data | Contour Options*.
2. In the *Contour Method* section of the *Contour Options* dialog, select the *Color Fill* option from the drop down list.
3. Click the *Color Ramp* button, and set the *Palette method* to *Hue Ramp*, and click *OK*.
4. Click *OK* to close the *Contour Options* dialog.
5. Choose *Display | Display Options* or right click in the tree to bring up the mesh display options. Select the *2D Mesh* tab if it is not already selected. Make sure that the *Contours* checkbox is checked so that the contours will display.
6. Click the *OK* button.

The display will refresh with color filled contours such as those shown in [Figure 8-13](#).

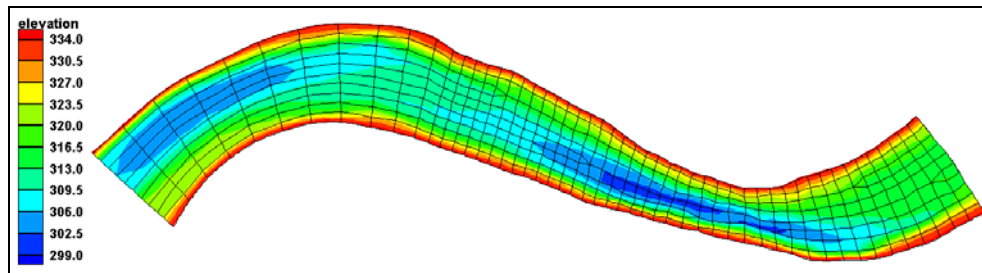


Figure 8-13 Elevation contours of the poway1 mesh.

In this plot, you can see that there are two pits in the river, while both banks are the highest part. If your contours are displaying red in the pits and blue along the banks, you can reverse the color ramp to match that of **Error! Reference source not found.**

1. Choose *Data | Contour Options*.
2. Select the *Color Ramp* button and then click the *Reverse* button at the bottom of the *Color Options* dialog.
3. Click *OK* and then *OK* again to exit *Display Options*.

For more examples of how to work with display and contour options in *SMS*, see the *SMS Help*.

8.10 Checking the Mesh Quality

Another important thing to check with a newly created finite element mesh is the mesh quality. There are various things that *SMS* looks at when checking this. To turn on the mesh quality:

1. Select *Display | Display Options* or right click in the *Project Explorer* to bring up the mesh display options.
2. Select the *2D Mesh* tab if it is not already selected.
3. Uncheck the *Contours* box.
4. Check the *Mesh quality* box.
5. Click the *OK* button.

The display will refresh without contours and with the mesh quality, as shown in [Figure 8-14](#). Mesh quality shows where problem areas may occur. A legend shows the color corresponding with each quality item. See the *SMS Help* for more information on these mesh quality options.

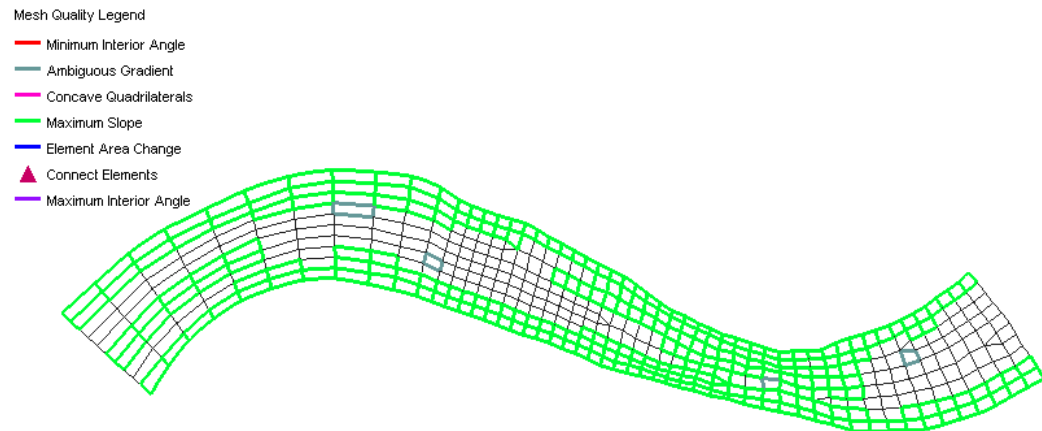


Figure 8-14 Mesh quality for the Poway1 finite element mesh.

Many elements are highlighted because of the maximum slope warning. Elements that are steep in the flow direction may cause supercritical flow to occur. They also could be related to an area where the depth averaged flow assumptions are invalid. In this mesh, however, the elements are steep in the direction perpendicular to the flow. This is a potential area of numerical instability if drying will take place. If a node on an element that spans a large range of elevation dries, all the flow through that element must be redistributed to other elements. In this case the entire mesh will be wet, so this warning can be ignored. To turn off this mesh quality check:

1. Bring up the display options.
2. Select the *2D Mesh* tab if it is not already selected.
3. Click the *Options* button next to the *Mesh quality* item.
4. In the *Element Quality Checks* dialog, turn off the *Maximum slope* option.
5. Click the *OK* button in both dialogs.

Once again, the display will refresh (see [Figure 8-15](#), but this time, no slope warnings will be shown. There are only *Ambiguous Gradient* warnings left for four elements, which are shown in the figure

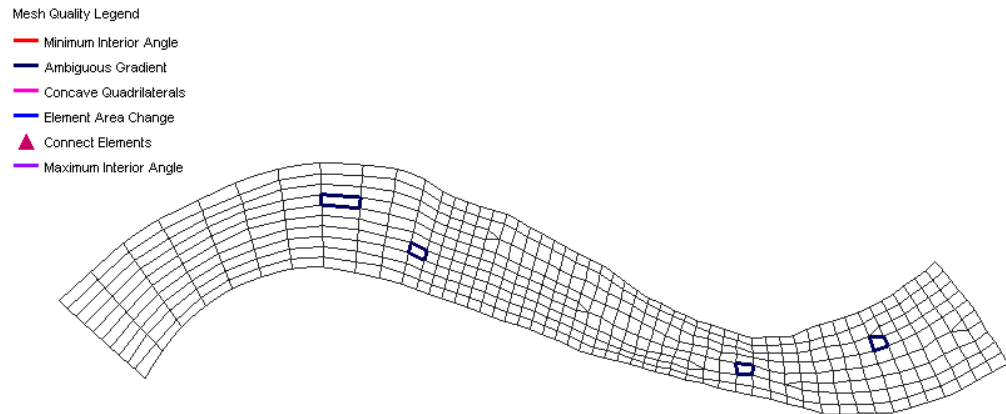


Figure 8-15 Mesh Quality without the Maximum Slope quality check.

If the ambiguous gradient is very small, it can be ignored because the surface is really almost planar. This is the case with the two ambiguous gradients on the left and the rightmost case. Ambiguous gradient cases with a larger variation in elevation, such as the one in the middle of this case, should just be split into two triangular elements.

The ambiguous gradients can be examined by selecting the nodes on the corners of the elements and viewing the elevation of these nodes.

After making these modifications, you do not need to worry about the element quality warnings. The following three things should be done (in no particular order):

- *Turn off the display of element quality checks.* You are done looking at the mesh quality, so this should be turned off to make the screen less busy.
- *Turn on the display of color filled contours.* Check that the adjustments you made did not make funny looking contours in the mesh. When editing nodal elevation values, it is always important to check the contours. If funny looking contours result, you may want to put things back the way they were and make some different changes.
- *Renumber the mesh.* Remember, whenever you adjust the finite element mesh, it should be renumbered. If you had only modified elevation values, then the mesh would not require renumbering. However, you must renumber the mesh after splitting a quadrilateral into two triangles.

8.11 Refining Elements

At times, it is desirous to refine part of a mesh so that there is more definition in that area. More definition helps to increase accuracy and decrease divergence problems. It is important to not refine a mesh too much, however, because more nodes and elements increase the time required for finite element computations. In this section, you will refine the section of elements on the left edge of the mesh.

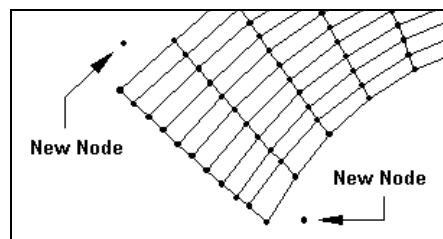
8.11.1 Inserting Breaklines

The elements at the left of the mesh are already rather skinny. The first refinement will be to cut them across their width. This can be done using a nodestring as a breakline. As you have seen previously, a nodestring must be created using existing nodes. Therefore, before being able to create a nodestring to cut the elements, you must create two nodes, one on either side of the channel. To do this:

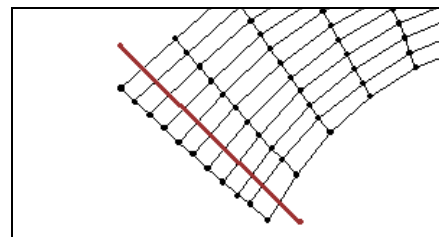
1. Zoom into the area shown in [Figure 8-16](#)
2. Choose the *Create Mesh Node*  tool from the *Toolbox*.
3. Click once on each side of the channel, near the middle of the left-most column of elements, as shown in **Error! Reference source not found.a**. After creating the first node, assign the z-value in the edit window to be 335.0. This will assign the elevation of the new nodes to be similar to the existing nodes.

A nodestring can now be created from one of these new nodes to the other. This nodestring will be used as a breakline. To create the nodestring:

1. Choose the *Create Nodestring*  tool from the *Toolbox*.
2. Click on one of the new nodes. Double-click on the other. The nodestring will appear, as shown in [Figure 8-16b](#).



(a). Two nodes to create.



(b). The nodestring to create.

Figure 8-16 -Adding the nodestring to use as a breakline.

With the nodestring created, it can be used as a breakline. A breakline splits all the elements that it crosses, forcing element edges to appear along the line. To make a breakline from the nodestring:

1. Choose the *Select Nodestring*  tool from the *Toolbox*.
2. Select the icon that appears in the center of the nodestring.
3. Select *Nodestrings | Force Breaklines*. The elements will be split along the nodestring, as shown in [Figure 8-17](#).

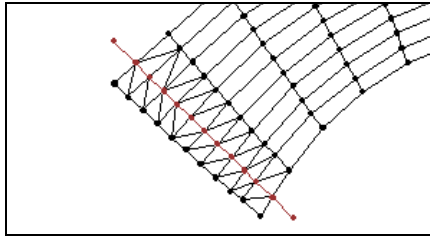


Figure 8-17 The breakline has been inserted.

Now that the nodestring has been used as a breakline, it is no longer needed. It should still be selected. To remove the nodestring:

- Select *Edit | Delete* or click the *Delete*  macro.

When the elements get broken along the breakline, triangular elements are created. These should be merged into quadrilateral elements. To do this:

1. Choose the *Select Element*  tool from the *Toolbox*.
2. Select *Edit | Select With Poly*. This allows you to select a specific set of elements by drawing a polygon around them.
3. Click out a polygon that surrounds all the triangular elements that were created by the breakline. Double-click to end the polygon.
4. With the triangular elements are highlighted, select *Elements | Merge Triangles*.
5. Refresh the display.

All of the triangular elements that were created by the breakline will be merged into quadrilateral elements. With these elements created, you just need to get rid of the two nodes that were created to define the breakline. These nodes are not connected to any elements, and are thus called *disjoint*. To remove the disjoint nodes:

1. Select *Nodes | Select Disjoint*. You should get a message that two disjoint nodes were found and selected. Click *OK* to this prompt.
2. Select *Edit | Delete* or the delete key.

Now that the breakline has been inserted, triangular elements have been merged into quadrilaterals, and the disjoint nodes have been deleted, the mesh should look like that in [Figure 8-18](#).

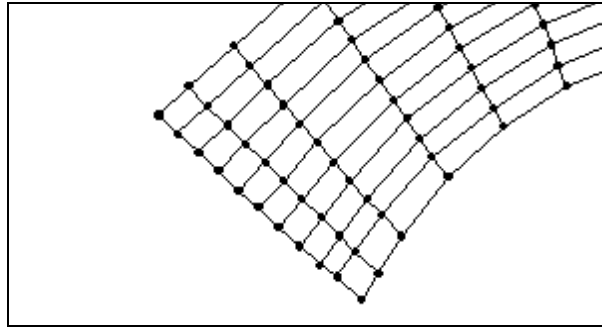
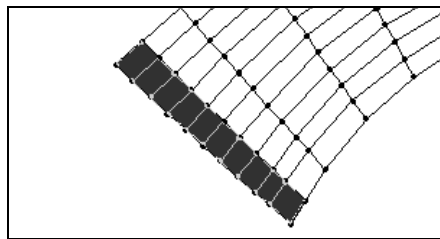


Figure 8-18 The final mesh after inserting the breakline.

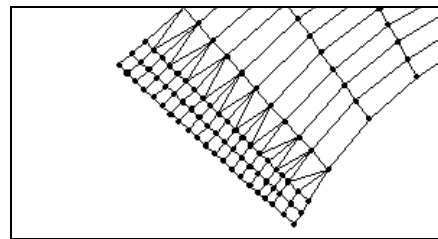
8.11.2 Using the Refine Command

Now that the breakline has been inserted, you are ready to use the refine command. This command splits a quadrilateral element into fourths. The reason the breakline was added is so that the refined elements would not be too skinny. You will refine the first column of elements on the very left side. To refine these elements:

1. Choose the *Select Element*  tool from the *Toolbox*.
2. Hold the *CTRL* key and drag a line through the left-most column of elements, as shown in [Figure 8-19a](#).
3. Select *Elements | Refine*. Each of the selected quadrilaterals will be split into four smaller quadrilaterals, and triangles will transition these small quadrilaterals to the larger quadrilaterals, as shown in [Figure 8-19b](#).



(a). The elements to select.



(b). After the refine command.

Figure 8-19 The section of the mesh to refine.

8.12 Finishing the Mesh

Now that elements have been created and edited, the following things should be done before using this mesh in a finite element analysis:

- The *Mesh Quality* should be checked. You will see the same types of warnings as in section 8.10. (You may want to turn off contours if they are on since the display could become cluttered.)

- The mesh should be renumbered. Remember that whenever nodes and elements are created, the mesh order should be fixed as in section 8.8.

8.13 Saving the Mesh

If SMS is registered, then the finite element mesh can be saved. This mesh will not be used in other tutorials, so saving it is not required. To save the mesh:

1. Select *File | Save As*.
2. Make sure the *Save as type* is set to *Project Files*.
3. Enter the name *poway1* and click the *Save* button.

8.14 Conclusion

This concludes the [Mesh Editing](#) tutorial. Although not every option was discussed, you should be familiar with many of the tools that SMS provides for mesh editing. You may continue to experiment with the SMS interface or you may quit the program. To quit SMS at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

Basic RMA2 Analysis

9.1 Introduction

This lesson will teach you how to prepare a mesh for an *RMA2* simulation. You will be using the project file *stmary.sms*. This project contains a simulation (“*.sim*” file) for *RMA2* similar to the result created in lesson 2. The file needed for this tutorial can be found in the *tutorial/tut08* directory. The simulation includes links to all the files needed by *RMA2* (or *TABS-MD*) to run an analysis. The actual input data is stored in the files named in the simulation file. To open the file:

1. Select *File | Open*.
2. Open the file *stmary.sms* from the *tutorial/tut08* directory.
3. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The mesh that is read in includes geometry (nodes and elements), material properties, and boundary conditions.

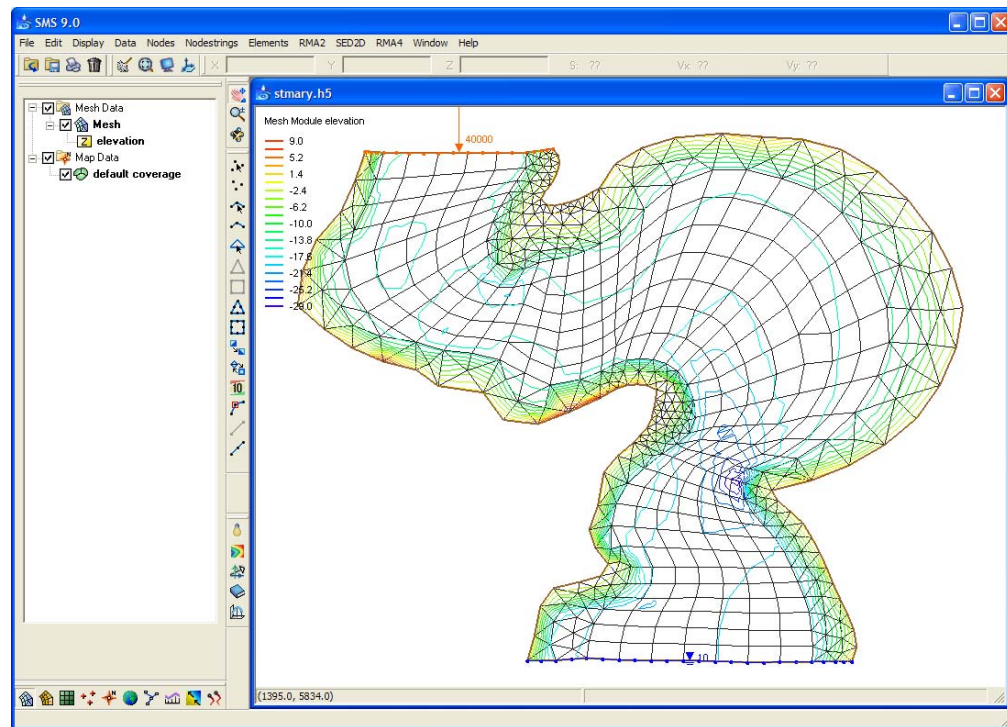


Figure 9-1. The mesh from stmary.sms.

9.2 Defining Material Properties

Each element of the mesh is assigned a material type. Each material type includes a value for Manning's roughness coefficient, parameters for turbulence, and parameters for wetting and drying. These material properties must be changed for this analysis. The material properties define how water flows through the element (see the *SMS Help* for details of what each parameter represents). To edit the material parameters:

1. Select *RMA2 | Material Properties*.
2. In the *RMA2 Material Properties* dialog, highlight the material *main_channel* (ID 1).
3. Under the *Turbulence* tab, make sure the *Standard eddy viscosity method* option is turned on and check the *Isotropic values* box. Enter a value of 50 for the eddy viscosity (E_{xx}). its hard to see the subscripts
4. Under the *Roughness* tab, specify a roughness value (n) of 0.03.
5. Highlight the material *left_bank* (ID 3). Set E_{xx} to 50 and set n to 0.045.
6. For the material labeled *right_bank* (ID 2), Set E_{xx} of 100 (higher turbulence requires a higher viscosity value) and set n to 0.04.

7. Click *OK*.

The material properties have now been properly defined.

Note: the material zones can be displayed by opening the *Display Options* dialog and turning on the *Materials* option under the *2D Mesh* tab.

9.3 Model Parameters

RMA2 includes many model parameters that may be set to represent various conditions. These include physical attributes such as water temperature and density, weather conditions such as wind, general material properties, and numeric controls. These are set in the *RMA2 | Model Control* command. For this simulation, we will use the default values. If you want to examine these:

1. Select *RMA2 | Model Control*.
2. Peruse through the tabs looking at the options.
3. Click the *OK* button when you are done.

9.4 Saving the Simulation

The boundary conditions (inflow rate and head at the outflow) were previously defined inside the map module. These were read in with the simulation. The entire simulation can now be saved. To save the simulation:

1. Select *File | Save As*.
2. Make sure the *Save as type* is *Project Files* and enter the name *stmary_ready.sms*.
3. Click the *Save* button to save the simulation.

9.5 Running the Simulation

To run the simulation:

1. Select *RMA2 | Run RMA2*.

This command actually performs several tasks. These include:

1. Performing a model check to detect missed components. If no problems are detected, this step produces no visible effects. If the model is missing a

required component (for example, if no boundary conditions existed), or there is an error in the simulation (such as an invalid mesh domain), a list of problems is posted for the user.

2. Running the Geometry File Generation (*GFGEN*) program. Before running the finite element analysis, the ASCII geometry file created by *SMS* must be converted to a binary format that *RMA2* can understand. The program is launched from automatically when the user runs the simulation. The location of the *GFGEN* executable is stored as a model preference. The progress of *GFGEN* will be displayed in a *Model Wrapper* dialog.
3. Running the *RMA2* simulation program. Once the binary geometry file is generated, the *Model Wrapper* dialog waits for the user to move on to the actual simulation. The location of the *RMA2* executable is also stored as a model preference. The progress of the model is displayed in the *Model Wrapper* dialog. Some of this paragraph seems redundant

For this simulation, *RMA2* should finish quickly. The *Model Wrapper* dialog waits for the user to acknowledge the completion of the model run. By default, it will then load the solution file when you click the *Exit* button. (If you are running in *Demo Mode*, the solution *stmary_ready.sol* is found in the *tutorials/tut08/output* directory and can be opened with the *File|Open* command.)

With the solution loaded, you are ready to evaluate the results. To do this:

1. Open the *Display Options*  dialog.
2. Under the *2D Mesh* tab, check the *Contours* and *Vectors* options (if not already turned on).
3. Under the *Contour Options* tab, select *Color Fill* as the *Color Method*.
4. Under the *Vectors* tab, select *Scale length to magnitude* as the option for *Shaft Length*.
5. Close the *Display Options* dialog.

The *RMA2* solutions for velocity magnitude, water depth and water surface elevation can be viewed by selecting the desired data set in the *Project Explorer*.

9.6 Conclusion

This concludes the Basic *RMA2* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

RMA2 Incremental Loading

10.1 Introduction

This lesson will teach you how to use revision cards for a spin down simulation in *RMA2*. The geometry has already been created and renumbered. To open the file:

1. Select *File | Open*.
2. Open the file *tribbase.geo* from the *tutorial\tut09* directory. If you have mesh data open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The geometry data will open, as shown in Figure 10-1.

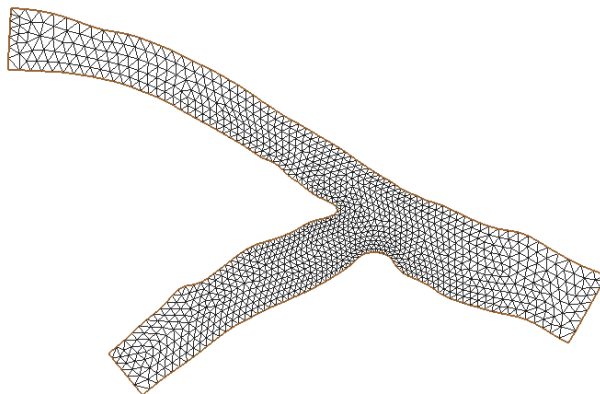


Figure 10-1. The mesh contained in the file *tribbase.geo*.

10.2 Specifying Model Units

Before continuing, make sure that the units are English. To do this:

1. Select *Edit | Current Coordinates*.
2. Make sure the *Horizontal System* is set to *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*.
3. Click *OK* to exit the dialog.

10.3 Defining Model Parameters

Several model control parameters must be assigned to define the state of the model. These model parameters include items such as how to handle wetting and drying, the model units, simulation time, and the number of iterations to be performed by *RMA2*. Additional information on these parameters is found in the *SMS Help* and the *RMA2* documentation. To define the model parameters:

- Select *RMA2 | Model Control*.

This opens the *RMA2 Model Control* property sheet, in which the model parameters are controlled. The rest of this section describes some of the property sheet tabs.

10.3.1 General

The *General* page contains various items such as simulation titles that describe what is being modeled. To set a title:

1. Click the *General* tab.
2. Enter the text “RMA2 Incremental Loading Tutorial” into the *Title1* field.

The remaining data fields on the *General* tab can be left as their default values. The machine type is used by *RMA2* to set numeric precision. The temperature and density parameters are fairly self explanatory. The section containing scale factors tells *RMA2* to scale the geometry, but is not used much since the geometry can be scaled inside of *SMS*. The initial water surface defaults to the elevation of the highest node in the mesh..

10.3.2 Timing

The *Timing* page contains options for defining model run time, iterations and convergence. This run involves a steady state (non-transient) simulation, so only the applicable parameters are discussed here. To set these values:

1. Click the *Timing* tab.
2. Be sure the *Simulation Type* is set to *Steady state*.
3. In the *Iterations For Flow Calculations* section, set the number of *Initial solution* iterations to 20. The model will run this number of iterations unless it converges first.
4. In the *Depth Convergence Parameters* section, set the *Steady state depth convergence* value to 0.0002. The model has converged when the maximum change in water depth between iterations at each node is less than this value.

10.3.3 Files

The *Files* page is used to specify various file options. To set these:

1. Click the *Files* tab.
2. Turn on the *Specify geometry file* option. Enter the file name “tribmesh.geo” and click the *Save* button. This allows a single geometry file to be used with multiple RMA2 simulations to avoid re-writing copies of the geometry in each simulation.
3. Turn on the *Write hotstart file* option to have RMA2 save a hotstart output file from this simulation that can be used in other simulations.

10.3.4 Materials

The *Materials* page is used to set items relating to material properties such as default roughness, eddy viscosity method and wetting/drying. To set these parameters:

1. Click the *Materials* tab.
2. In the *Global Roughness Assignment* section, set the *Default roughness value* to 0.03.
3. Be sure the *Global Eddy Viscosity Assignment* type is set to *Traditional eddy viscosity approach*. With this method, exact eddy viscosity values need to be assigned for each material.

To accept all the above values:

- Click the *OK* button to close the *RMA2 Model Control* dialog.

10.4 Defining Boundary Conditions

For this tutorial, flowrate and water surface elevation will be defined along nodestrings at the *open boundaries* of the mesh. An open boundary is a boundary where water is allowed to enter or exit. Generally for *RMA2*, a flowrate is specified across inflow boundaries and water surface elevation is specified across outflow boundaries. Other available boundary conditions are rating curves and reflecting boundaries.

This model has two inflow and one outflow boundaries so three nodestrings must be created. The upper tributary is the main river and has much more water than the lower tributary. These boundaries are highlighted in Figure 10-2.

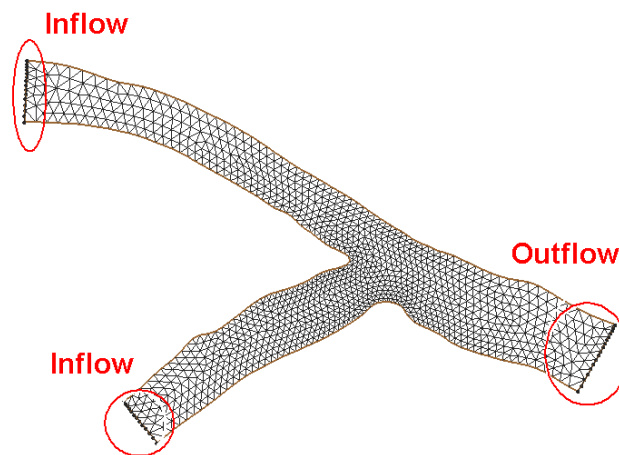


Figure 10-2. Position of the boundary nodestrings in the mesh.

10.4.1 Creating Nodestrings

Nodestrings should be created from right to left when looking downstream and the first nodestring should be that which spans across the whole river section. Therefore the outflow nodestring at the right should be created first. To create this:

1. Choose the *Create Nodestrings* tool  from the *Toolbox*.
2. Start the nodestring by clicking on the lower node at the outflow boundary.
3. Hold the *SHIFT* key and double-click on the upper node at the outflow boundary to create and end the nodestring.

The inflow nodestrings can be created now. To do this:

- Create a nodestring across the top inflow boundary and then across the bottom one. Create each right to left when looking downstream.

10.4.2 Defining Flow Boundary Conditions

To assign the first flow condition:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring.
2. Select the top left nodestring by clicking on the icon.
3. Select *RMA2 | Assign BC*.
4. Change the *Boundary Condition Type* to *Specified Flowrate* and assign a constant *Flowrate* of 55,000 (cfs).
5. Make sure the *Flow Direction* is set to *Perpendicular to Boundary*.
6. Enter a *Flow Distribution* of 0.750. This allows more flow to come into the mesh in the deeper areas.
7. Click the *OK* button to assign the boundary condition.

This defines the top left nodestring to be an inflow boundary condition. To define the second inflow boundary condition:

1. Select the bottom left nodestring.
2. Repeat the above steps to assign a perpendicular flow of 600 (cfs).

10.4.3 Defining Head Boundary Conditions

A water surface elevation (head) boundary condition will be assigned to the outflow boundary nodestring. To assign this boundary condition:

1. Select the outflow nodestring.
2. Select *RMA2 | Assign BC*.
3. Change the *Boundary Condition Type* to *Water surface elevation* and assign a constant value of 33 (feet).
4. Make sure the *Make this nodestring the "Total Flow" nodestring* option is checked because this boundary is the combination of both tributaries.
5. Click the *OK* button to assign the boundary condition.

10.5 Defining Material Properties

Each element in the mesh is assigned a material type ID. This particular geometry has two material types. To see each of these materials:

4. Select *Display | Display Options* or click the *Display Options*  macro.
5. In the *2D Mesh* tab, turn on the *Materials* option.
6. Turn off the *Nodes* and *Elements* options.
7. Click the *OK* button to close the *Display Options* dialog.

The display should look something like Figure 10-3. Most of the model is made of material one but there are spots with a second material type.

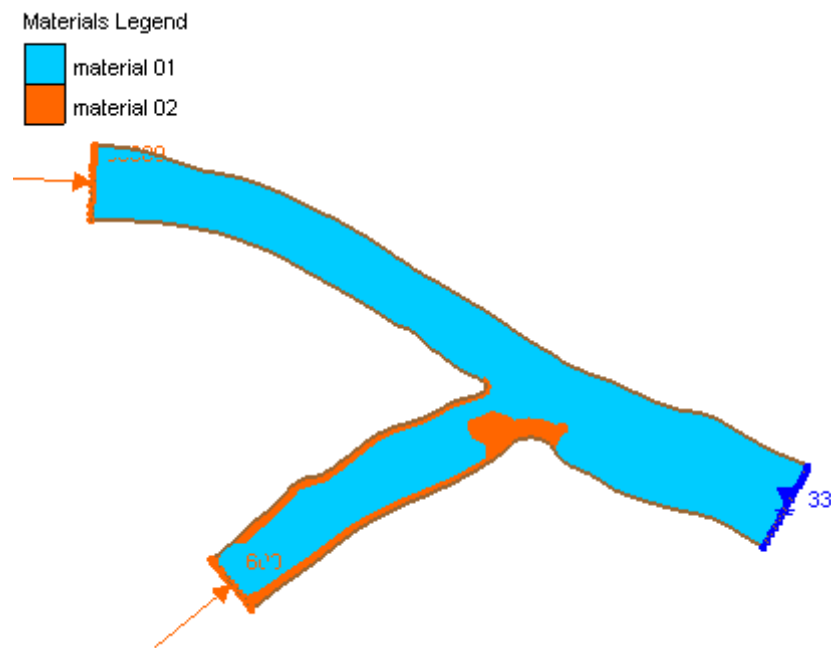


Figure 10-3. The display of materials.

Before continuing, turn off the material display. To do this:

- Open the *Display Options* dialog. Turn off the *Materials* option and turn the *Elements* options back on.

The materials were created with default parameters that must be changed for this particular simulation. The material properties define how water flows through the element. To edit the material parameters:

1. Select *RMA2 | Material Properties*.

2. In the *RMA2 Materials Properties* dialog, highlight *material 01*.
3. Under the *Turbulence* tab, make sure the *Standard eddy viscosity method* option is selected and the *Isotropic Values* box is checked. Enter a value of 25 for the eddy viscosity (E_{xx}) for this material.
4. Under the *Roughness* tab, make sure the *Override global specification* option is turned off for this material. This will use the global value of 0.03 that was already set in the *Materials* page of the *RMA2 Model Control* dialog.
5. Highlight *material 02* and set isotropic eddy viscosity to 50. Override the global roughness value for this material and set n to 0.04.
6. Click the *OK* button to close the *RMA2 Material Properties* dialog.

The eddy viscosity and roughness parameters have now been defined for this model.

10.6 Saving The Simulation

SMS can launch the *RMA2* numerical model. However, the model will read the data from files. Therefore, the simulation must be saved prior to running. If you haven't done this, you will be prompted to do so. To save the simulation:

1. Select *File | Save As*.
2. Change the *Save as type* to *TABS Simulation (*.sim)* and enter "trib" as the *File name*.
3. Click the *Save* button to save the simulation.

10.7 Running the Model

At this point, you are ready to try running *RMA2*. To do this choose *RMA2 | Run RMA2*. Before *SMS* launches the model, a quick check is done on the data to make sure everything is valid. This model check will bring up the dialog shown in Figure 10-4 if any anomalies are detected. For this model, two warnings should be detected.

The first warning says that the default starting elevation for the simulation will leave portions of the domain dry. This can lead to instabilities. One option is to raise the initial water level, but if it is much higher than the outflow boundary elevation, instabilities can develop at those locations. If the simulation is not stable with the default elevation, it may be best to use incremental loading as will be demonstrated later. For now we will ignore this warning.

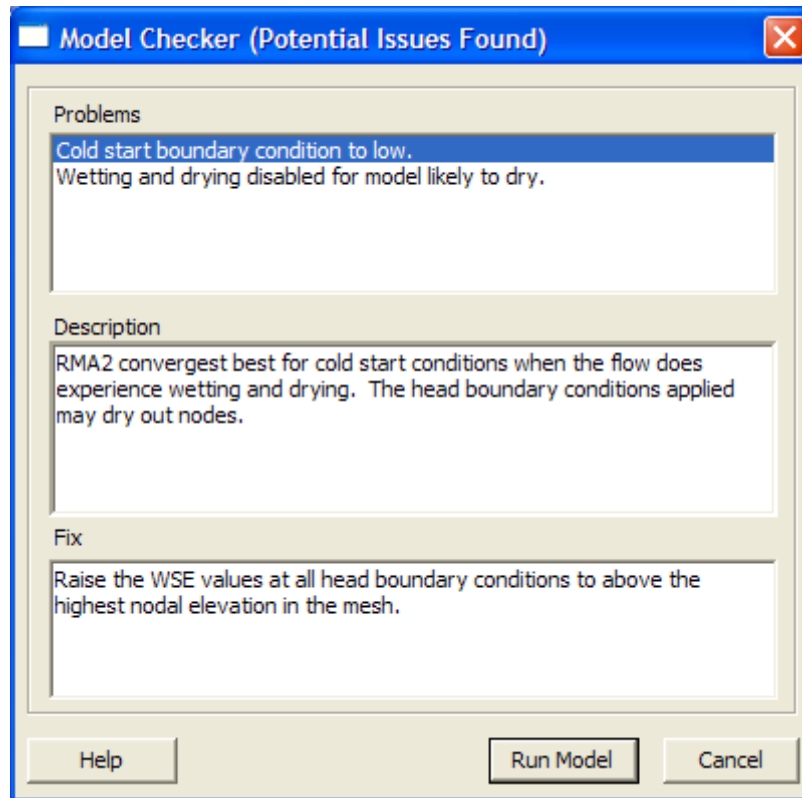


Figure 10-4. Warning in RMA2 data.

The second warning says that the elements might dry out, so the wet/dry flag should be turned on. To do this:

1. Select the *Cancel* button to leave the *Model Checker* without running RMA2.
2. Select *RMA2 | Model Control*.
3. On the *Materials* page turn on the *Turn on wet/dry check* option.
4. Click *OK* to close the *RMA2 Model Control* dialog.
5. Click the *Run Check* button again to assure that the second error has gone away. Then click the *Done* button to close the *RMA2 Model Checker*.

Because there is still a warning message, RMA2 might not converge. However, before attempting to fix this, the simulation will be tried.

10.8 Running RMA2

6. Since this is a new RMA2 simulation, SMS will first run the *GFGEN* program.

7. In the *Run Model* dialog, SMS shows the location of the *GFGEN* executable (gfgv435.exe). Click the *OK* button to launch *GFGEN*.
8. When *GFGEN* finishes, click the *Run RMA2* button.
9. In the *Run Model* dialog, SMS shows the location of the *RMA2* executable (rma2v435.exe). Click the *OK* button to launch *RMA2*.

RMA2 will run through three iterations and diverge. At the bottom of the *RMA2* output window is the text “STOP depth convergence exceeds 25.0”, as highlighted in Figure 10-5. This is how *RMA2* declares that it has not successfully converged.

- Click the *Exit* button to close the *RMA2* output window.

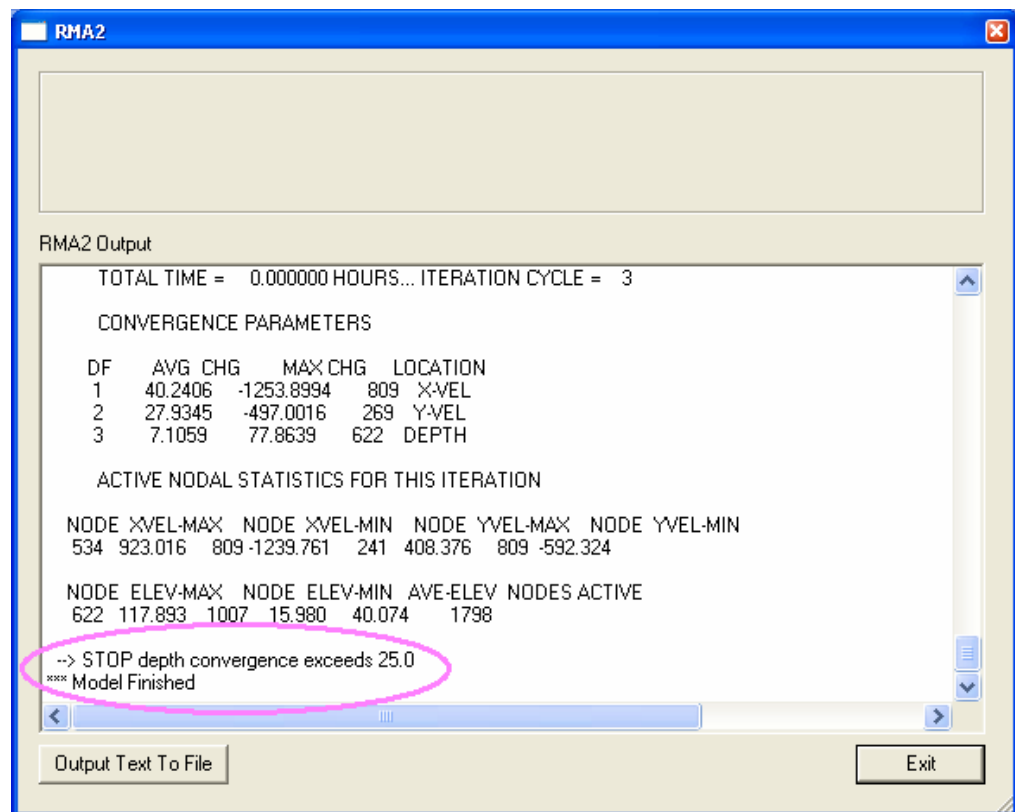


Figure 10-5. Output from running *ld0.sim*.

10.9 Using RMA2 Revisions

Various things can contribute to a model not converging. In this case, SMS had given an error message that the initial water surface was low. The low water surface elevation for this simulation does not allow *RMA2* to converge from the coldstart simulation. Revisions can be defined to allow the simulation to converge. A set of

revisions is like a set of hotstart simulations except that they are all run at once with no user-interaction.

10.9.1 Checking For High Bathymetry

The first thing to do is find the highest bathymetric value from all nodes. To do this:

1. Select *File* | *Get Info* or click the *Get Info*  macro.
2. In the *Mesh Info* tab, look at the *Max Z value*, highlighted in Figure 10-6.

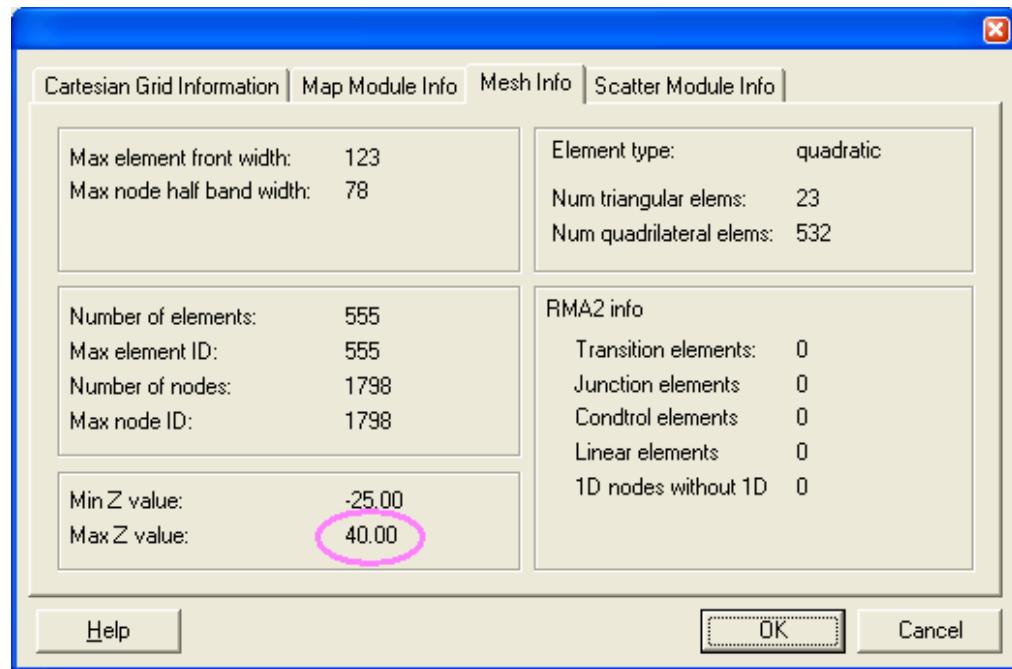


Figure 10-6. Mesh Information dialog.

10.9.2 Changing Initial Water Surface Elevation

The initial water surface elevation value should be increased to be above the highest bathymetry value. To change this value:

1. Select  the outflow node string.
2. Select *RMA2* | *Assign BC*.
3. Change the *Water Surface Elevation* value to 41 (feet) and click OK.

10.9.3 Creating Revisions

The water surface elevation can be revised to the final value of 36 feet through a series of revisions. To create the revisions:

1. Make sure the outflow nodestring is still selected.
2. Select *RMA2 | Revisions*.
3. In the *Existing Revisions* section, right-click the “0.0 hours” text and choose the *New Revision* item.
4. In the Selected Revision section, change the type to *Nodestring BC*.
5. Click the *Add* button and set the *Water Surface Elevation* value to 40 (feet). Then click the *OK* button in the *RMA2 Assign Boundary Conditions* dialog.
6. Repeat the above steps to add four revisions, revising the boundary condition to 39, 38, 37 and finally 36 feet.
7. Click the *Close* button to assign these revisions.

RMA2 will first run an initial simulation with the downstream water surface elevation at 41 feet, then will run five additional simulations, revising the boundary condition down by one foot each time. Each successive step will use the previous solution as a starting point for the next simulation. Only one solution file will be generated and it will have the final water surface elevation value of 36 feet.

10.9.4 Running a New Simulation

You can now try running this simulation with the set of revisions. To do this:

1. Select *File | Save As*.
2. Set the *Save as type* to *TABS Simulation (*.sim)* and save “ld1.sim”.
3. Select *RMA2 | Run RMA2*. You should not need to re-run *GFGEN*.
4. Click the *OK* button in the *Run Model* dialog. *RMA2* should successfully run the initial simulation and all five revisions.
5. When *RMA2* finishes, click the *Exit* button.

The solution can be opened analyzed using the options described in the Post-Processing tutorial.

10.10 Conclusion

This concludes the RMA2 tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.



Figure 10-7

SED2D-WES Analysis

This lesson will teach you how to use *RMA2* in conjunction with *SED2D-WES* to perform sediment transport simulations. The files required for this tutorial are 's.sim', 's.bc', 's.geo', 's_gf.run', 's_rm.run', and 'hydrograph.xys'. The *RMA2* simulation will model a 5,000 cms hydrograph through a simple 'S' channel bend with a base flow of 3,000 cms.

SED2D-WES requires the hydrodynamics to be computed elsewhere and given as part of its input. This is usually done with another *TABS* component, *RMA2*. An underlying assumption of *SED2D-WES* is that the bed does not change enough to significantly alter flow velocities. When large changes in the bed occur due to either erosion or deposition, a new hydrodynamic solution should be computed before continuing the *SED2D-WES* simulation.

Currently, there is no support in *SED2D-WES* for US units. Metric units should be used for all data, including the geometry and hydrodynamic simulation. Although it is possible to define conversion factors from US units for the geometry and the hydrodynamics, the SMS developers do not recommend such practice.

11.1 TABS Data Flow

In order to be successful at modeling with *SED2D-WES*, you need to understand the data flow through the *TABS* models. There are many files that are used for this type of modeling situation and it is easy to get lost in all the data. Figure 11-1 shows the most important files that are involved in the *SED2D-WES* analysis process. The files in this image are described below.

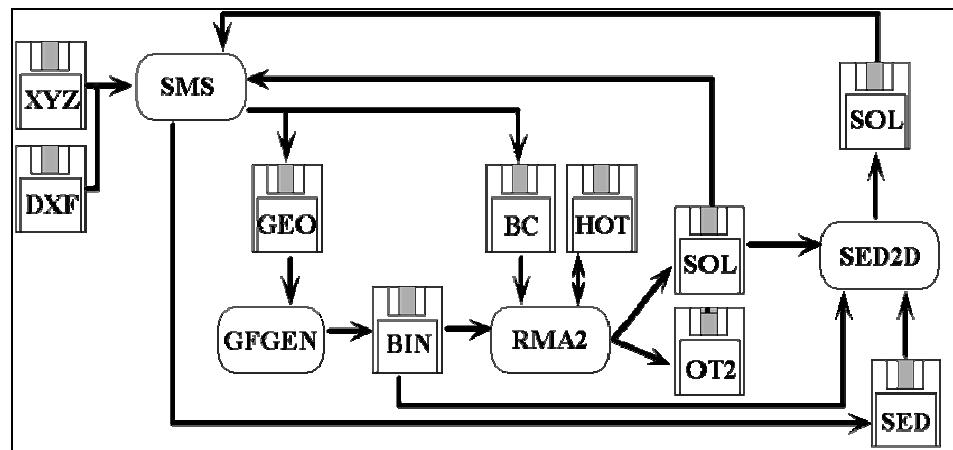


Figure 11-1. The TABS models data flow.

SMS saves three types of data that are required to perform a *SED2D-WES* simulation. These are the geometry (.geo) file, hydrodynamic boundary conditions (.bc) file, and sediment transport definition (.sed) file. The geometry is created by combining location and bathymetric data from a variety of sources such as surveys, dxf files, and USGS quad sheets. After the geometry is built, appropriate boundary conditions are applied to define the hydrodynamic and sediment simulations.

11.1.1 GFGEN

The geometry file saved by SMS is written in ASCII format. The TABS component *GFGEN* converts the ASCII geometry file into a binary geometry file (.bin). This binary geometry file is used by all other TABS modeling components to define the model domain.

11.1.2 RMA2

After the binary geometry file has been created, it can be used as input, along with the hydrodynamic boundary conditions, to generate a hydrodynamic solution, or flow field, over the model domain. Either a steady state or dynamic flow field can be used with *SED2D-WES*. When using a dynamic flow field, an initial steady state flow field should be generated, using the boundary conditions that will be applied to the first time step of the dynamic simulation. This additional steady state simulation can be used to generate a hotstart of suspended sediment concentrations before running the *SED2D-WES* analysis.

A basic assumption of *SED2D-WES* is that the flow field will not significantly change with small amounts of deposition/erosion. At the end of each time step, *SED2D-WES* saves a new ASCII geometry file (_out.geo), which has the accumulation of deposition or erosion added into the bathymetry of each node. At times, large changes in the bed occur, which require the flow field to be recomputed with a new geometry. By default, *SED2D-WES* stops when the deposition or erosion

at any node exceeds 25% of the original water depth. If such a condition occurs, the *RMA2* simulation should be restarted where *SED2D-WES* stopped, using the updated geometry to compute a more accurate flow field. Using the new flow field, the *SED2D-WES* simulation can be continued.

11.1.3 SED2D-WES

Part of the *SED2D-WES* simulation definition includes a user-assumed suspended sediment concentration over the entire model domain. As the simulation starts, the initial concentration at each node is quickly adjusted, based on computations of the governing equations. Such adjustments cause an artificially high deposition/erosion rate until the suspended sediment concentrations settle down to model-expected values. To keep these adjustment effects out of the final simulation, *SED2D-WES* should be run using an initial steady state hydrodynamic solution and constant sediment boundary conditions equivalent to the desired starting condition. The solution generated can be used to hotstart the suspended sediment concentrations.

11.2 RMA2 Coldstart Simulation

The hydrograph to be used with the *SED2D-WES* simulation is accompanied by a rating curve to define the downstream water surface. Before running the hydrograph analysis, a steady state solution will generate initial velocity and suspended sediment concentration values.

- Open the simulation *s.sim* from the *session6* directory.

This simulation has the geometry and material properties set up. To find out more about setting up these entities, see the tutorials that discuss *RMA2* simulations. The geometry has a constant slope of 0.002. The upstream boundary is at the top left cross section while the downstream boundary is at the bottom right cross section.

As was previously stated, the downstream water surface will be determined using a rating curve. However, it is often difficult to run a coldstart simulation with a rating curve. Before running the steady state rating curve simulation, a coldstart simulation will be run using a head boundary condition on the downstream end.

From the rating curve, a flowrate of 3000 cms produces a downstream water surface elevation of about 10.0 meters. Notice, however, that the highest nodal bathymetry value is above 18.0 meters. An *RMA2* coldstart simulation must start with the downstream water surface set higher than all the node bathymetry values or the simulation most likely will not converge. This downstream head for this coldstart simulation will be set to start at 30 meters and slowly lower to 10 meters.

To set up global parameters for this initial steady state simulation:

1. Select *RMA2 | Model Control* to open the *RMA2 Model Control* dialog.

2. Select the *Files* tab and turn on the *Write hotstart file* option.
3. Because this tutorial uses multiple simulations without changes to the nodes or elements data, we want to avoid re-writing multiple geometry files. Turn on the *Specify geometry file* option and select to always save the geometry as “s.geo”. Click *Yes* to confirm.
4. Select the *Timing* tab and make sure the *Steady State* option is selected.
5. Set the *Iterations* for the *Initial Solution* to 100 and set the *Steady state depth convergence* to a convergence value of 0.0005.
6. Select the *General* tab and turn on the *Specify the initial water surface* option. Change the value to 30.0 (meters).
7. Click *OK* from the *RMA2 Model Control* dialog.

Setting the number of iterations to a very high value and turning on the convergence parameter assures that *RMA2* converges before continuing to the next step. If *RMA2* does not reach convergence, the solution is invalid. The initial water surface elevation value on a coldstart simulation should be assigned to be close to the downstream head value. It is applied to all nodes in the mesh as a starting point so that a solution can be obtained.

To define the initial boundary conditions:

1. Create a nodestring  across the upstream (top left) boundary. Be sure to create it from right to left while looking downstream.
 2. Select the nodestring. Arrows should appear, pointing into the mesh. If the arrows point upstream, it was not created in the correct direction and you should choose the *Nodestring | Reverse Direction* menu command.
 3. Select *RMA2 | Assign BC*. Choose the *Specified Flowrate* option and assign a *Flowrate* of 3000 (cms) and keep the *Flow Direction* set to *Perpendicular to boundary*. Then click *OK*.
 4. Create a nodestring across the downstream (bottom right) boundary. Once again, be sure to create it from right to left when looking downstream.
 5. Select the nodestring. This time, be sure the arrows point out of the mesh.
 6. Select *RMA2 | Assign BC*. Select the *Water surface elevation* option and assign an *Elevation* value of 30 (m). Then click *OK*.
- The initial simulation for a constant downstream head of 30 meters has now been defined. However, we want to progressively lower this boundary condition to 10 meters. This can be done in *RMA2* using revisions, which can be created inside

SMS. Revisions to a boundary condition can be defined within SMS as long as the appropriate nodestring is selected. To add the revisions:

1. With the bottom right nodestring still selected, choose *RMA2 | Revisions*.
2. Because this is a steady state model, only one time step is shown. Right-click on time zero and select *New Revision*.
3. At the bottom right of the dialog, select *Nodestring BC* from the dropdown menu and then click the *Add* button.
4. For this first revision, change the water surface elevation to 27 (meters) and click the OK button to make the assignment.
5. Create eight more revisions by repeating steps 2-4 above, but set the water surface elevation for each revision as indicated below.

Revision 2	24 (meters)
Revision 3	22
Revision 4	20
Revision 5	18
Revision 6	16
Revision 7	14
Revision 8	12
Revision 9	10

These revisions cause RMA2 to run a series of steady state simulations, each time slightly lowering the water surface elevation. It is like running nine hotstart simulations, without having to ever interrupt RMA2 in the process. In the end, you will end up with only one solution having the water surface elevation assigned as the final revision value.

With the revisions added to the boundary conditions, you are ready to run the initial solution. The new simulation must first be saved. To save the simulation:

1. Select *File | Save As*.
2. From the *Save as type* list choose “Tabs Simulation (*.sim)”.
3. Enter the name “s-cold” and click the *Save* button.

Now that the model is saved, you can run the simulation. To do this:

1. Select *RMA2 | Run RMA2*. You should be shown the location of the gfgv435.exe file to first run *GFGEN*. Note: if you see the message “gfgv435.exe - <not

found>”, click the *File Browser*  button and locate it (*GFGEN* should be in the *models* directory under the SMS installation).

2. Click the *OK* button to run *GFGEN*.
3. When *GFGEN* is finished, click the *Run RMA2* button. You will see a dialog showing the location of *RMA2*. Find the executable if necessary and run the simulation.
4. When *RMA2* is finished, click the *Exit* button (it takes about 1.5 minutes on an Intel 450 MHz system to execute all revisions).

RMA2 creates a solution named “s-cold.sol”. If you want, you can open this and verify that the downstream water surface elevation has been revised down to 10 meters. But the solution will not be used for anything else. The component that we will need is the file “s-cold.hot”, which is a hotstart file with the downstream water surface elevation of 10 meters. Future runs will use this hotstart simulation.

11.3 RMA2 Rating Curve Simulation

The hotstart file saved from the previous *RMA2* run needs to be used as input to the rating curve simulation. To do this:

1. Select *RMA2 | Model Control* to open the *RMA2 Model Control* dialog.
2. Select the *Files* tab, and turn on the *Hotstart input file* option.
3. Open the file “s-cold.hot”.
4. Click the *OK* button from the *RMA2 Model Control* dialog.

You are now ready to introduce the rating curve into the simulation. The rating curve definition consists of four values, which describe an exponential equation to compute the flowrate, and a fifth value to define the flow direction (see the *BRC* card in the *RMA2* help file).

The boundary conditions created by *SMS* are an upstream flowrate and downstream head. The head boundary needs to be converted to a rating curve. To add the rating curve:

1. Select the downstream nodestring (bottom right – it may still be selected).
2. Choose *RMA2 | Assign BC*, and change the *Boundary Condition Type* to *Rating Curve*.
3. Click the *Options* button and enter the values of $A1=1440$, $A2=500$, $E0=8$, and $c=1.0$. Then click *OK* from the *Rating Curve Options* dialog.

4. A rating curve requires a specified flow direction. The default (in the flow direction field) is perpendicular into the mesh but because this is an outflow condition, you need to change it to be out of the mesh. In the *Flow Direction* section, turn off *Perpendicular to boundary* and change the angle to 270.
5. Click *OK* to exit the *RMA2 Assign Boundary Conditions* dialog.

These define an initial set of rating curve values that was found to be compatible with the coldstart simulation. However, these are not the final desired values. You must use a series of revisions to reach the final desired values.

When you changed the downstream boundary condition type from water surface elevation to rating curve, SMS automatically removed the revision data. But the revisions themselves are still there. To modify the revision data:

1. With the downstream nodestring still selected, choose *RMA2 | Revisions*. If you click the plus sign at time zero you can see that the revisions are still defined but they have no data because the boundary condition type changed.
2. Select the first revision. Choose the “Nodestring BC” item from the drop-down box and click the *Add* button.
3. Click the *Options* button next to *Rating Curve* and change the A2 value to 550 and the A1 value to 1450. All other values should remain the same as the original boundary condition specification.
4. Click the *OK* button to close both the *Rating Curve Options* and the *RMA2 Assign Boundary Conditions* dialogs.
5. Repeat steps 2-3 for revisions 2, 3, 4, 5 and 6, each time setting the A1 value to 1450 and the A2 values to 600, 625, 650, 675 and finally 700.
6. Delete revisions 7, 8 and 9 by right-clicking each and choosing *Delete*.
7. Click the *Close* button to close the *RMA2 Revisions* dialog.

Now that the outflow boundary condition and revisions have been updated, you are ready to run this second simulation. To do this:

1. Select *File | Save As*.
2. Once again change the *Save as type* box to “*Tabs Simulation (*.sim)*” and then *Save* the simulation as “s-init.sim”.
3. Select *RMA2 | Run RMA2*. Since the geometry has not changed, you should not need to rerun GFGEN. Click *OK* to run *RMA2*.
4. When *RMA2* is finished, click *Exit* (it takes about 3 minutes on an Intel 450 MHz system to execute all revisions).

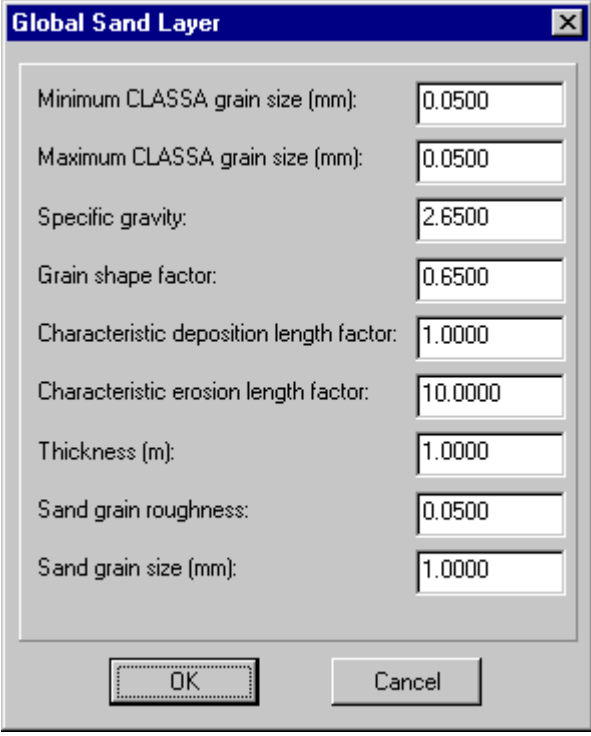
RMA2 creates a solution named “s-init.sol”, which will be used to define the flow field for the initial steady state *SED2D-WES* simulation. In addition, *RMA2* creates the file “s-init.hot”, which is a hotstart file for the full hydrograph simulation with a downstream rating curve.

11.4 Initial SED2D-WES Simulation

You now have a steady state *RMA2* solution whose boundary conditions are the same as the base flow condition for a hydrograph simulation. Before running the *RMA2* hydrograph analysis, you should run a *SED2D* simulation using this steady state flow field to generate initial suspended sediment concentrations throughout the mesh.

For this *SED2D-WES* simulation, a constant incoming suspended sediment concentration of 0.5 ppt (parts per thousand) will be used. The bed type is fine sand with an effective grain size of 0.05. To set up the bed information:

1. Select *SED2D | Global Parameters* to open the *SED2D Global Parameters* dialog.
2. Make sure the *Bed Type* is set to *Sand* and click the *Set Up Bed* button. Set the values as shown in Figure 11-2 and click the *OK* button. Note: for the current version of *SED2D-WES* (3.2), the *Minimum* and *Maximum* CLASSA grain sizes must be equal.
3. Set the *Diffusion coefficients* to 1.5. This determines how fast the sediment in suspension is distributed throughout the model.
4. Assign the *Initial concentration* as 0.5. This defines the initial suspended sediment concentration at all nodes in a *SED2D-WES* coldstart simulation. It is analogous to the *Initial WSE* value in *RMA2* to define the initial water surface elevation at all nodes of an *RMA2* coldstart simulation.
5. Change the *Settling velocity* to 0.001 (m/s).
6. Click the *OK* button to close the *SED2D Global Parameters* dialog.

A screenshot of a software dialog box titled "Global Sand Layer". It contains nine input fields with numerical values and two buttons at the bottom: "OK" and "Cancel".

Parameter	Value
Minimum CLASSA grain size (mm):	0.0500
Maximum CLASSA grain size (mm):	0.0500
Specific gravity:	2.6500
Grain shape factor:	0.6500
Characteristic deposition length factor:	1.0000
Characteristic erosion length factor:	10.0000
Thickness (m):	1.0000
Sand grain roughness:	0.0500
Sand grain size (mm):	1.0000

Figure 11-2. The Sand bed values to assign.

The next step to define the *SED2D-WES* simulation is to set up the equation and time controls. To do this:

1. Select *SED2D | Model Control* to open the *SED2D Model Control* dialog.
2. Set the *Hydraulic Bed Shear Stress* to use *Manning equation*.
3. Assign the *Time step length* as 0.5 hours, *Simulation time* as 24.0 hours, and the *Number of cycles* as 48 (maximum number of time steps to run).
4. Click the *OK* button to close the *SED2D Model Control* dialog.

The final part of setting up the *SED2D-WES* simulation is to assign the suspended sediment concentration at all inflow boundary conditions. This concentration is specified in ppt, parts per thousand. To assign the suspended sediment concentration:

1. Select the inflow nodestring and choose *SED2D | Assign BC*.
2. Assign a *Constant* concentration of 0.5 (ppt) and click *OK*.

The *SED2D-WES* simulation is now ready to be saved. To do this:

- Select *File | Save RMA2*.

The SED2D data is added into the s-init simulation. With the initial simulation saved, you can run *SED2D-WES*. This simulation uses the binary geometry and solution files that were created in the previous section. To run this simulation:

1. Select *SED2D | Run SED2D* and click the *OK* button to run *SED2D*. Once again, if the location of *sed2dv32.exe* is unknown, find it manually.
2. Click the *OK* button (it takes about 45 seconds on an Intel 450 MHz system). When *SED2D-WES* finishes, press the *Exit* key in its window.

The file “s-init_dbed.sol” is created and will be used to hotstart the suspended sediment concentrations for the hydrograph analysis. Note that this is the *SED2D-WES* solution file and it can be opened inside *SMS*, but this would not have significant meaning at this point of the analysis.

11.5 RMA2/SED2D Hydrograph Analysis

You have created an *RMA2* solution that has base flow conditions before the hydrograph is introduced. You also have a *SED2D-WES* solution that has suspended sediment concentrations for the *RMA2* solution. You are now ready to introduce the hydrograph and obtain the dynamic flow field solution so that you can run the true *SED2D-WES* simulation.

11.5.1 Updates For RMA2

The first thing to do is update the *RMA2* model controls. To do this:

1. Select *RMA2 | Model Control* to open the *RMA2 Model Control* dialog.
2. Select the *Timing* tab and change the *Simulation type* to *Dynamic*.
3. Set the *Time step size* to 0.5 (hours) and the *Number of time steps* to 48 so that the *RMA2* simulation models one full day. Also, set the *Maximum time* to 24.0 (hours).
4. Increase the *Iterations for Each time step* to 100 and set the *Dynamic depth convergence* to 0.0005 to assure convergence of *RMA2* at each time step.
5. In the *Files* tab, change the *Hotstart input file* to “s-init.hot” instead of “s-cold.hot”.
6. Click the *OK* button to accept these changes.

Now that a dynamic simulation is set up, you need to assign a hydrograph at the upstream boundary. A time series file was previously saved which defines the hydrograph values for various hour intervals. To assign this boundary condition:

1. Select the upstream flow nodestring and choose *RMA2 | Assign BC*.
2. Change the flowrate type to *Transient*.
3. Click the curve button. In the *XY Series Editor* dialog, click the *Import* button and *Open* the file “hydrograph.xls”. Then click *OK*. The assigned hydrograph is shown on the curve button.
4. Click the *OK* button to assign the hydrograph boundary condition.

The downstream boundary condition must be changed to be equal to the values from the last revision in the s-init simulation. To change this boundary condition:

1. Select the rating curve nodestring and choose *RMA2 | Assign BC*.
2. Click the rating curve *Options* button. Change the A1 value to 1450 and the A2 value to 700, then click the *OK* button.
3. Click the *OK* button to assign these new rating curve values.

The last piece of information that must be updated is the set of revisions. Because we have a hotstart condition at our desired starting point, we no longer need any revisions. To delete all the revisions:

1. Select *RMA2 | Revisions*. Notice that the dialog now lists each time step in the dynamic simulation. There is only a plus next to hour zero because only it has any assigned revisions.
2. Right-click on hour zero and choose *Delete All Revisions*.
3. Click the *Close* button to accept these changes.

11.5.2 Updates for SED2D

The previous *SED2D-WES* data should still be set up. However, the previous simulation did not use an input hotstart file for suspended sediment concentrations. You need to tell *SED2D-WES* that this simulation does use an input hotstart file. To do this:

1. Select *SED2D | Global Parameters* to open the *SED2D Global Parameters* dialog.
2. Under the *Hotstart Options* section, turn on *Concentrations*, then click the *File Browser*  button and *Open* the file “s-init_dbcd.sol”.
3. Click the *OK* button to close the *SED2D Global Parameters* dialog.

11.5.3 Saving the Final Simulation

To save this new simulation:

1. Select *File* | *Save As* (do not select *Save Project*).
2. Change the *Save as type* to “TABS Simulation (*.sim)”.
3. Enter the name “s-hydro” and click the *Save* button.

11.5.4 Running the Simulation

After saving the project and making the required changes to the saved data files, you are ready to run the simulation. To do this:

1. Select *RMA2* | *Run RMA2*.
2. Select *RMA2* | *Run RMA2* and press the *OK* button to run *RMA2* (it takes about 9.5 minutes on an Intel 450 MHz system to execute all time steps). When *RMA2* finishes, click on *Exit*.
3. After running *RMA2*, you can run *SED2D*. Select *SED2D* | *Run SED2D* and press the *OK* button to run *SED2D-WES*. When *SED2D-WES* finishes, press the *Exit* key in its window.

After running the simulation, there are two solution files that can be opened into *SMS*: the hydrodynamic *RMA2* solution “s-hydro.sol” and the sediment transport *SED2D-WES* solution “s-hydro_dbed.sol”. In addition, *SED2D-WES* saves a new ASCII geometry file named “s-hydro_out.geo”, which contains updated bathymetry values at all nodes in the mesh. This is in the same format as the geometry file originally saved by *SMS*.

- Open the s-hydro.sol and s-hydro_dbed.sol solution files. Look at the contours from these simulations.

For tips and assistance on viewing and interpreting the simulation results, see the tutorial lessons that discuss post-processing techniques.

11.6 Conclusion

This concludes the *SED2D-WES* Analysis tutorial. You may continue to experiment with *SMS* or you may close the program. To exit *SMS* at this time:

- Select *File* | *Exit*. Click the *Yes* button if asked to confirm.

RMA4 Analysis

12.1 Introduction

This lesson will teach you how to run a solution using *RMA4*. If you have not yet completed Lesson 4 on *RMA2*, you should do so now. *RMA4* is part of the TABS-MD suite of programs and is used for tracking constituent flow in 2D models. In this lesson, you will use *RMA4* to model 3 situations: an inflow of a constituent into a river, the inflow of a constituent into a bay, and salinity intrusion.

12.2 Case 1

RMA4 can only be run after having initially run a solution in *RMA2*. This is because *RMA4* uses the flow solutions computed by *RMA2* to compute the constituent concentration as it flows through the mesh. An *RMA2* geometry and solution have been supplied. To open the *RMA2* files:

1. *Select File | Open.*
2. Select the file *madora.spr* from the *tutorial\tut10* directory. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button.

The geometry will be displayed on the screen with the *RMA2* boundary conditions.

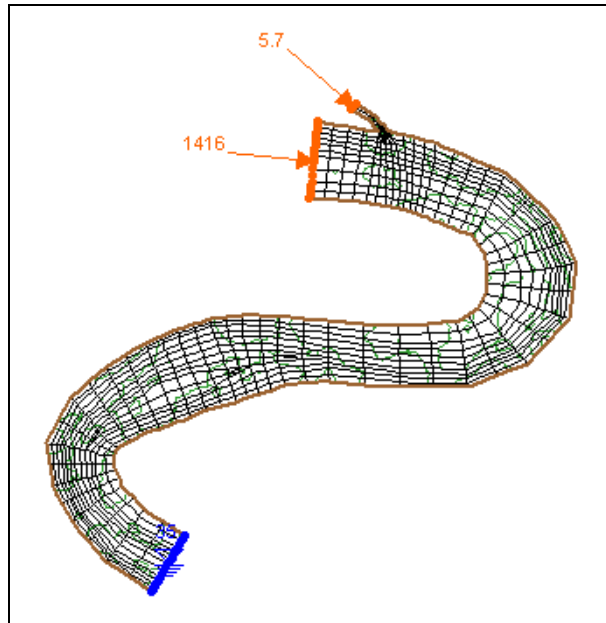


Figure 12-1 Madora mesh

The mesh was created in Metric units because *RMA4* requires Metric units. The main channel has a flow of 1,416 cms (50,000 cfs) with a channel entering with a flow of 5.7 cms (2,000 cfs).

12.2.1 RMA4 Model Control

RMA4 is a transient model. The *RMA2* solution is a steady state solution. *RMA4* will assume a steady flow throughout the mesh, but the boundary conditions will change. To set the time that *RMA4* will run:

1. Go to *RMA4 | Model Control*.
2. Set the times as follows:
 - *Start time*: 0.0
 - *Time step*: 0.5 (hrs)
 - *Total time steps*: 50
 - *Max time*: 24 (hrs)
 - *Last time step from the RMA2 velocity file to be used*: 0.0 (hrs)
 - *Number of hours subtracted from the RMA2 velocity file*: 0.0 (hrs)

3. Make sure the *Geometry input file*, *Flow input file*, *Binary output file*, and *Full output options* are on and the other *File Control* options are off.
4. Push the *Advanced Output Control* button, turn the *Print all* command on, and push *OK*.
5. Push *OK* to exit the *RMA4 Control* dialog.

12.2.2 Boundary Conditions

For this model, a pollutant has been dumped into the smaller channel for three hours. The concentration of the pollutant in the stream is 1,000 ppm. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the smaller inflow boundary (labeled as 5.7).
3. If the arrows are not pointing into the larger channel select the *Nodestrings | Reverse Direction*.
4. Select *RMA4 | Assign BC*.
5. Switch to *Transient* and push the *Define Curve* button.
6. The *XY Series Editor* dialog appears. In this dialog, a time series curve can be created. To turn the pollutant on for only 3 hours:
 - a. Click the *New* button.
 - b. Enter the following *Time/Concentration* values:

Time	Concentration
0.0	1000.0
3.0	1000.0
3.1	0.0
24.0	0.0

- c. Push *OK* to exit the *XY Series Editor*.
7. Push *OK* to exit the *RMA4 Assign BC* dialog.

NOTE: In this case, we applied 1,000 ppm as a boundary condition. *RMA4* does not care about the units of the concentration. The output from *RMA4* is relative to the initial number you specify. For example, we specified a concentration of 1,000. The

values in the solution will range from 0 to 1,000 as the plume spreads downstream. We can say that the concentration was ppm, ppt, or kg/kg; RMA4 treats all concentrations as relative values.

12.2.3 Material Properties

The final step is to specify the Material diffusion. To do this:

1. Select *RMA4 | Material Properties*.
2. Select each material, turn on the *Material diffusion*, and set the *X* and *Y diffusion coefficients* to 10.0 (m²/s).
3. Push *Close* to exit the dialog.

Because *RMA4* does not have the ability to model turbulence, diffusion coefficients may be used to approximate turbulence. By assigning a diffusion coefficient in the *x* and *y* directions for each material, the flow over that material will be altered somewhat to provide an approximation of turbulent flow over that region. A value of -1.0 may be applied (this is the default) to allow normal flow over the material. Positive values provide turbulence. The higher the value, the greater the effect is.

12.2.4 Run RMA4

You are now ready to run *RMA4*. Before running the model, you will need to save the data:

1. Select *File | Save as*.
2. Set the file type to *Project Files* and enter *madora* as the filename.
3. Click *Save* to save the file.

NOTE: *RMA4* requires that the *RMA2* and *RMA4* filenames be the same, so you must save the project as *madora.spr*.

To run *RMA4*:

1. Select *RMA4 | Run RMA4*.
2. If the prompt shows a message that *RMA4* is *not found*, click the *File Browser*  button manually find the correct program executable.
3. Click the *OK* button to launch *RMA4*.

RMA4 will take a few minutes to run. The file *madora.qsl* is the solution file containing the constituent data at each node.

12.2.5 Film Loop

Once a solution has been created by *SMS*, you can use a number of features to view the results and adjust the model to better approximate the observed values. The easiest way to view the results from the *RMA4* solution is to use the film loop. Before using the film loop, the *RMA4* solution file must be imported:

1. Select *File | Open*.
2. Select *madora.qsl*, and push *Open*.

To create the film loop:

1. Select *Data | Film Loop* to bring up the Setup dialog.
2. Select *Create New Filmloop* and click on the folder icon to save this new loop as *smsloop.avi*
3. Make sure the *Scalar/Vector Animation* option is selected, then press *Next*
4. Turn on *Scalar Data Set*, set the *Number of Frames* to 47, and *Run Simulation Time* from 0.5 to 24.0. Press *Next*.
5. Click the *Display Options* button .
6. In the *Display Options*, turn off everything but *Elements* and *Contours*.
7. Select the *Contour Options* tab.
8. Select *Color fill* as the Contour Method, and *Specify a range* from a *Min* of 0.0 to a *Max* of 10.0. (This is done because when the stream flow enters the main channel, the concentration quickly drops to between 0.0 and 10.0 ppm.)
9. For the *Contour Interval*, select *Specified Interval* and set it to 0.5.
10. Push *OK* to get back to the *Film Loop* dialog.
11. Push *Finish* to start generating the film loop. Each frame of the film loop will be generated. After the film loop is done generating, a new window will come up to play the results.
12. Click the *Stop* button and exit the AVI application when you are finished watching the results.

You will notice how the concentration drops quickly as the pollutant enters the main channel. This occurs because the inflow from the small channel is 0.4% of the inflow from the main channel.

12.3 Case 2

In this case, a constituent is coming into Noyo Bay from a river. The files for this case can be opened by:

1. Select *File | Open*.
2. Select the file *noyo1.spr* from the *tutorial\tut10* directory. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button.

The geometry will be displayed on the screen with the *RMA2* boundary conditions.

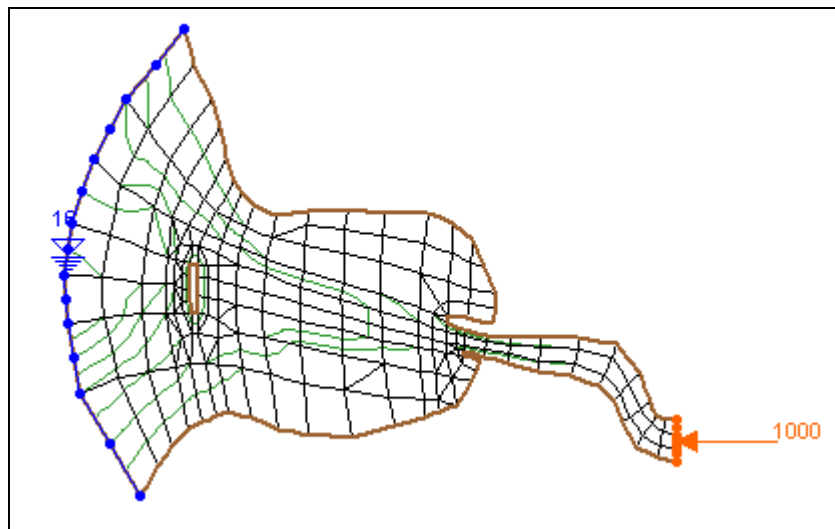


Figure 12-2 Noyo mesh

This mesh was created in English units. The river flowing into Noyo Bay has a flow of 1,000 cfs. The water surface elevation on the left side varies as the tide comes in and out over a 12-hour cycle repeated twice a day.

12.3.1 RMA4 Model Control

To set the model times:

1. Go to *RMA4 | Model Control*.
2. Set the times as follows:
 - *Start time*: 0.0
 - *Time step*: 0.5 (hrs)
 - *Total time steps*: 49

- *Max time:* 24 (hrs)
- *Last time step from the RMA2 velocity file to be used:* 24.0 (hrs)
- *Number of hours subtracted from the RMA2 velocity file:* 12.0 (hrs)

The last two times will cause *RMA4* to use the last 12 hours of the *RMA2* solution.

3. Make sure the *Geometry input file*, *Flow input file*, *Binary output file*, and *Full output options* are on and the other *File Control* options are off.
4. Push the *Advanced Output Control* button, turn the *Print all* command on, and push *OK*.
5. Push *OK* to exit the *RMA4 Control* dialog.

12.3.2 Boundary Conditions

For this model, a constant inflow of 100 ppm of a pollutant enters the bay from the river. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the right side of the model and select *RMA4 | Assign BC*.
3. Set a *Constant Concentration* of 100.0 (ppm) and push *OK* to exit the *RMA4 Assign BC* dialog.

12.3.3 Material Properties

To apply the diffusion:

1. Select *RMA4 | Material Properties*.
2. Select each material, turn on the *Material diffusion*, and set the *X* and *Y diffusion coefficients* to 1.0 (m²/s).
3. Push *Close* to exit the dialog.

12.3.4 Run RMA4

Save and run *RMA4* as you did for Case 1. Save the project as *noyo1.spr*. As mentioned earlier, the Noyo mesh was run using English units and *RMA4* requires metric units. When you save the file, *SMS* checks to see what the current units are set

as, which are English in this case. If the units are English, SMS writes the HS and GS cards to the .trn file to tell RMA4 to convert to metric units.

Run *RMA4* as done earlier. After *RMA4* runs, open the solution file *noyo1.qsl*.

12.3.5 Film Loop

Generate a film loop using the same steps as for case 1 (Section 12.2).

12.4 Case 3

For this final case, we will view salinity intrusion into Noyo Bay. Open *noyo2.spr*.

12.4.1 RMA4 Model Control

To set the model times:

1. Go to *RMA4 | Model Control*.
2. Set the times as follows:
 - *Start time*: 0.0
 - *Time step*: 0.5 (hrs)
 - *Total time steps*: 49
 - *Max time*: 24 (hrs)
 - *Last time step from the RMA2 velocity file to be used*: 24.0 (hrs)
 - *Number of hours subtracted from the RMA2 velocity file*: 12.0 (hrs)

The last two times will cause *RMA4* to use the last 12 hours of the *RMA2* solution.

3. Make sure the *Geometry input file*, *Flow input file*, *Binary output file*, and *Full output options* are on and the other *File Control* options are off.
4. Push the *Advanced Output Control* button, turn the *Print all* command on, and push *OK*.
5. Push *OK* to exit the *RMA4 Control* dialog.

12.4.2 Boundary Conditions

For this model, a constant concentration of 8 ppm exists offshore and enters the bay from the left. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the **left** side of the model and select *RMA4 | Assign BC*.
3. Set a *Constant Concentration* of 8.0.
4. Select the *Factor applied when flow direction changes* button and set the *Shock Factor* to 0.5.
5. Push *OK* to exit the *RMA4 Assign BC* dialog.

Since a concentration in water is rarely rigidly maintained, a shock factor may be applied to allow fluctuation of the concentration when the flow direction changes. If no shock factor is applied, no matter how much the flow pushes the concentration out of the model, the concentration at the boundary will not change. However, applying a shock factor is like creating a buffer zone outside the model where the constituent can go until the flow begins to carry it back into the model. This provides for a more realistic solution in some cases. Depending on the situation, a different shock factor may be applied from zero for no shock to 1.0 for a gradual change due to a change in flow direction.

12.4.3 Material Properties

To apply the diffusion:

1. Select *RMA4 | Material Properties*.
2. Select each material, turn on the *Material diffusion*, and set the *X* and *Y diffusion coefficients* to 1.0 (m²/s).
3. Push *Close* to exit the dialog.

12.4.4 Run RMA4

Save and run *RMA4* as you did for Cases 1 and 2. Save the project as *noyo2.spr*.

Run *RMA4* as done earlier. After *RMA4* runs, open the solution file named *noyo2.qsl*.

12.4.5 Film Loop

Generate a film loop using the same steps as for case 1. The only difference will be in the *Film Loop Options* dialog (after pushing *Setup*):

- In the bottom right of the dialog, set the *Run simulation from time* to 6.0 and the *Run simulation to* 18.0 hours.

Running these times will show a full tidal cycle that runs continuously.

12.5 Other Changes

You may want to play with the shock factor and diffusion coefficients to see how they affect the model. Other options include:

- Change the diffusion coefficients in all 3 cases to 0.5 and then try 10.0 to see the differences.
- Change the shock factor in the third case to 0.0 and 1.0. There is a large difference in how far the intrusion gets into the bay.

12.6 Conclusion

This concludes the *RMA4* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

HIVEL Analysis

13.1 Introduction

This lesson will teach you how to prepare a mesh for and run a solution using *HIVEL2D*. The files used by the simulation are referenced through a “.sup” file. The file *proto.sup* contains a link to the finite element mesh for the analysis and can be found in the *tutorial\tut13* directory. To open the file:

1. Select *File | Open*.
2. Find and highlight the file *proto.sup*. If you still have geometry data open from a previous tutorial, you will be warned that all existing mesh data will be deleted. If this happens, click *OK* to the prompt. The geometry data will open as shown in Figure 13-1.

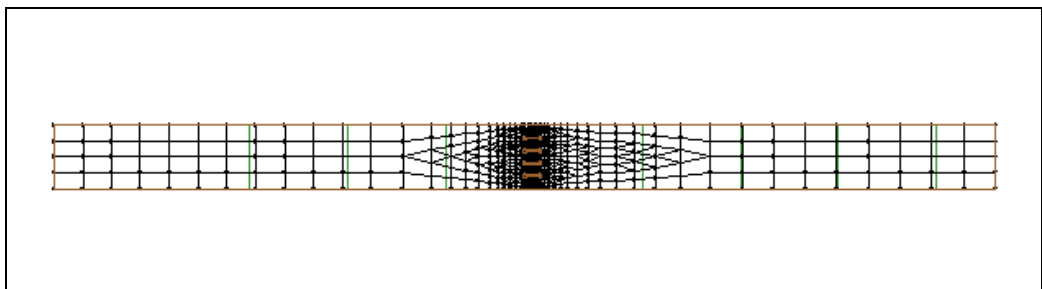


Figure 13-1 The mesh contained in the file *proto.geo*.

13.2 Creating Materials

Each element in the mesh includes a material type ID. The materials were created with default parameters that must be changed for this particular mesh. For *HIVEL* the material properties define the roughness for the elements.

There is only one material in this example. To edit the material parameter:

1. Select *HIVEL | Material Properties*.
2. In the *HIVEL Material Properties*, select the material labeled *material 01*.
3. Enter 0.014 for *n* (Manning's roughness).
4. Click the *OK* button.

The material now has the correct parameter associated with it. (Materials can be displayed by opening the *Display Options* dialog and turning on the *Materials* toggle under the 2D Mesh tab.)

13.3 Creating the Hotstart File

HIVEL2D must have an initial hotstart file to run a solution. *SMS* allows you to create this hotstart file either using constant values or a data set that has been previously loaded through the *Data Browser*. For this tutorial, a data set will be created using the *Data Calculator*.

1. Select *Data | Data Calculator*.
2. Enter "Start Depth" in the *Result* field.
3. Enter 7.22 in the *Expression* field. This is the constant depth.
4. Click the *Compute* button to add "Start Depth" to the Data Sets.
5. Click the *Done* button to leave the *Data Calculator* dialog.

Now the *HIVEL2D* hotstart file can be created.

1. Select *HIVEL | Build Hot Start*.
2. Click on the file icon  and enter the name and location for the hotstart file. You will tell *HIVEL* what file to use, so you can choose the name. By default the file name is *hivel.hot* and it will be created in the same area that you read the geometry from.
3. Enter 0.0 for the *Time associated with step m*.

4. For both *Step m-1* and *Step m*, select the *Constant* option for the discharge. Enter a value of 148.5 for p and 0.0 for q for both steps.
5. For *Step m-1*, select the *Data Set* option for the *Water Depth*. Press the *Select* button.
6. From the *Select Data Set* dialog highlight the “Start Depth” and press the *Select* button to exit.
7. Repeat steps 4 and 5 for *Step m*.
8. Push the *Write Hotstart Now* button. This writes the file and closes the dialog.

When *HIVEL2D* starts, it will use these values as an initial starting place. It is important to note that at the end of the computations, this file will be over written. Therefore, a backup copy may be created if desired. This is done by making a copy of the file using Windows Explorer.

13.4 Creating Nodestrings

Before boundary conditions can be applied at the inflow or outflow boundaries, nodestrings must first be created.

To create the nodestrings:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. To create the inflow nodestring, click on the node on the top left side of the mesh. While holding *Shift* key, double-click on the node on the bottom of the left side.
3. Repeat step 2 to create the outflow nodestring on the right side of the mesh.

13.5 Defining Boundary Conditions

13.5.1 General Parameters

The finite element network is only the first part of the numerical model. We have already defined the material properties associated with the elements in the mesh. In addition to the geometry and the material properties, we must define several other model parameters. To edit the *HIVEL2D* parameters:

1. Select *Edit | Current Coordinates* and make sure the *Horizontal System* is set to *Local* and both *Units* are set to *U.S. Survey Feet*. Click *OK*.

2. Select *HIVEL | Model Control*.
3. Click on the file icon in the upper portion of the left panel. This brings up a file selection dialog. Select the hotstart file (hivel.hot) that you just created.
4. Enter the following values:
 - Maximum number of iterations per time step = 6
 - Time step size = 4.0
 - Number of time steps = 100
 - Save output every = 100th time step
5. Push *OK* to exit the dialog.

HIVEL2D simulates steady state boundary conditions but uses pseudo-dynamic analysis to allow the flow to change from the initial conditions to the final boundary conditions.

13.5.2 Defining Steady State Flow and Head

For this tutorial, we will define boundary conditions at the nodestring of the inflow boundary, as well as along a nodestring at the outflow boundary.

To assign the inflow boundary condition:

1. Choose the *Select Nodestrings*  tool from the *Toolbox* and select the nodestring across the inflow boundary on the left side of the mesh.
2. Select *HIVEL | Assign BC*.
3. Select the *Inflow string* and *Supercritical* options.
4. Select *unit discharge*, *x and y components*, and *depth* for the *Inflow Parameters*.
5. Enter 148.5 for *P* and 0.0 for *Q* (x and y components of unit flow), and 7.22 (ft) for *depth*.
6. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

To assign the outflow boundary condition on the right side of the mesh:

1. Choose the *Select Nodestring*  tool from the *Toolbox* and select the nodestring across the outflow boundary.

2. Select *HIVEL* | *Assign BC*.
3. Select the *Outflow string* and *Supercritical* options.
4. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

For *HIVEL2D*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries for subcritical flow and at upstream boundaries for supercritical flow. If a jump occurs, you need to specify head at both boundaries. See the *HIVEL2D Reference Manual* for more about assigning boundary conditions.

13.6 Saving the Simulation

HIVEL2D uses a geometry file, boundary condition file, and hotstart file written by *SMS* to run an analysis. These files are specified in a *super file* that is also written by *SMS*. You must save data that has been created.

1. Select *File* | *Save As* and make sure the *Save as type* is set to *Project Files*.
2. Enter *proto_h* for the *File name* and press *Save*.

13.7 Using HIVEL2D

The analysis program *HIVEL2D* can be launched from inside *SMS*. To do this:

1. Select *HIVEL* | *Run HIVEL*.
2. *SMS* saves the location of the *HIVEL* executable as a preference. If this preference is defined, the model will launch. If the preference is undefined, *SMS* shows a message that the *hivel2d* executable is not found, click the *File Browser*  button to find the *HIVEL* executable and Click the *OK* button to run the model.

(Note:, when before *SMS* launches the model, it performs a model quality quick check. If any problems are detected a message box will be displayed for the user to respond to.)

HIVEL2D may take a few minutes to run this solution. When it is finished, the program may ask you to press *Return* before the window will go away. The model has simulated 400 seconds of flow time (100 time steps @ 4 seconds each). The run has created 3 new files:

- *proto_hflo.dat*: contains velocity data at each node.

- *proto_hwse.dat*: contains water surface elevation data at each node.
- *hivel.hot* – contains hotstart data to continue where *HIVEL2D* left off.

These files can be opened using the *File|Open* command. As each is read, a data set is added to the *Project Explorer* under the Mesh item.

13.8 Conclusion

This concludes the HIVEL Analysis tutorial. You may continue to experiment with the SMS interface or you may quit the program. To quit SMS at this point:

- Select *File | Exit*. If prompted to confirm, click the Yes button.

Basic FESWMS Analysis

14.1 Introduction

This lesson will teach you how to prepare a mesh for a *FESWMS* simulation. You will be using the project file *stmary.sms* which is similar to what you created in tutorial 2. The input file is included in the *tutorial\tut05* directory. This project file includes a *FESWMS* project (*.*fpr*) file. It contains a list of filenames that are used by *FESWMS*. The actual input data is stored in the files named in the project file. To open the file:

1. Select *File | Open*.
2. Open the file *stmary.sms* supplied in the *tutorial\tut14* directory. The display will refresh with the mesh as shown in Figure 14-1.

The mesh that is read in includes geometry (nodes and elements from a *.*net* file), as well as material properties and boundary conditions (from a *.*dat* file).

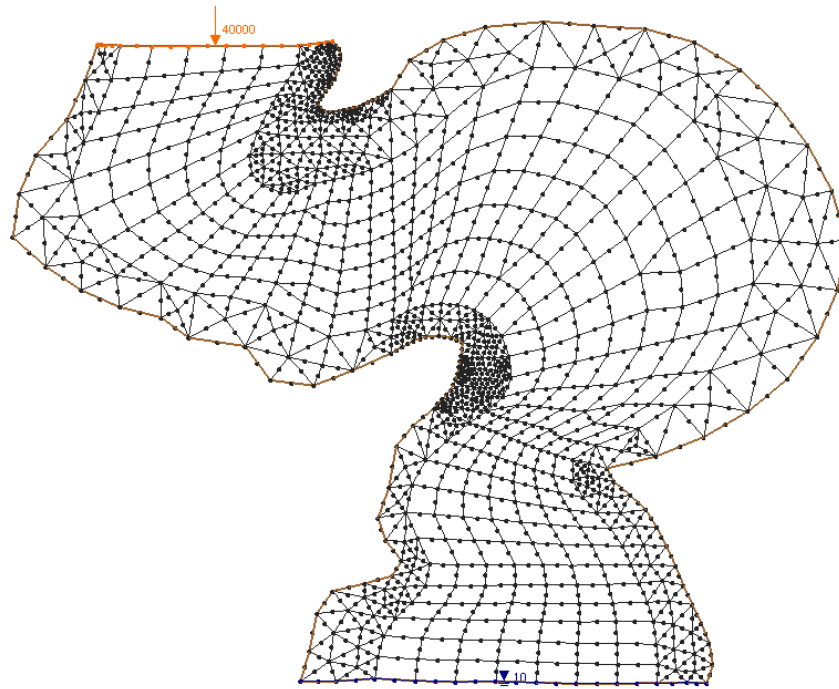


Figure 14-1. The mesh contained in *stmary.sms*.

14.2 Converting Elements

For *FESWMS*, it is best to use 9-noded quadrilateral elements (quads) even though both 8-noded and 9-noded quads are supported. The mesh generation process from the conceptual model generates 8-noded quads to increase compatibility. To convert these to 9-noded quads:

- Select *Elements* | *QUAD8* <-> *QUAD9*.

The screen will refresh and the quadrilateral elements will have 9 nodes. Since there was a change in the number of nodes, the mesh should be renumbered, even though it was renumbered before being saved. To do this:

1. Choose the *Select Nodestring*  tool from the *Toolbox*.
2. Click in the selection box at the downstream boundary condition (at the bottom of the screen).
3. Select *Nodestrings* | *Renumber*.

14.3 Defining Material Properties

Each element in the mesh is assigned a material type. Each material type includes parameters for roughness, turbulence, and wetting/drying. These material properties must be changed for this analysis. The materials properties define how water flows through the element (see the SMS Help for details of what each parameter represents). To edit the material parameters:

1. Select *FESWMS | Material Properties*.
2. In the *FESWMS Material Properties* dialog, select the material *main_channel*.
3. In the *Roughness Parameters* tab of the *FESWMS Material Properties* dialog, an image shows what the Manning's coefficients are for different depths. Enter 0.03 as the roughness value (n) for both depths. We are not dealing with scour in this problem, so we can ignore the other roughness values.
4. In the *Turbulence Parameters* tab of the *FESWMS Material Properties* dialog, enter a value of 50 for the kinematic eddy viscosity (V_o).
5. Highlight the material named *left_bank*. Set V_o to 50 and set n to 0.045.
6. For the material named *right_bank*. Set V_o to 100 (higher turbulence requires a higher viscosity value) and set n to 0.04.
7. Click the *OK* button to close the *FESWMS Material Properties* dialog.

The kinematic eddy viscosity and Manning's roughness values should always be set. Other material properties can also be set for more advanced problems. See the *FESWMS* documentation for more information on these other material properties.

Optional: The materials can be displayed by opening the *Display Options* dialog and toggling the *Materials* option on. If you do this, be sure to turn the option back off before continuing with this lesson.

14.4 Setting Model Parameters

Before running an analysis, model controls and parameters must be set. The parameters and files used are specified in the *FESWMS Control* dialog. To change the global parameters:

1. First, to set the units to English, go to *Edit | Current Coordinates*.

2. Make sure the *Horizontal System* is *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*. Press *OK* to exit the *Current Coordinates* dialog.
3. Select *FESWMS | Model Control*.
4. In the *FESWMS Version* section, make sure *FESWMS 3.** is selected.
5. In the *FST2DH Input* section, make sure the *NET file* option is the only box selected.
6. Make sure the *Solution Type* is set to *Steady state*.
7. Click the *Parameters* button and set the values shown below:
 - Water-surface elevation = 10.0
 - Unit flow convergence = 0.005
 - Water depth convergence = 0.001
 - Element drying / wetting = ONLeave the other defaults and push *OK*.
8. Click the *Iterations* button and set the number of *Iterations* to 10. Then click the *OK* button to return to the *FESWMS Control* dialog.
9. Click the *Print...* button and make sure the *ECHO to screen* option is turned on. Then click the *OK* button to return to the *FESWMS Control* dialog.
10. Click the *OK* button to exit the *FESWMS Control* dialog.

14.5 Saving the Simulation

The boundary conditions (inflow rate and head at the outflow) were previously defined using the conceptual model. These were read in with the simulation. The entire simulation can now be saved. To do this:

1. Select *File | Save New Project...*
2. Enter the name *stmary_ready.sms*, and push *Save*.

The model control options and boundary conditions are saved to the file *stmary_ready.dat*, and the finite element network is saved to the file *stmary_ready.net*. If desired, look at the file *stmary_ready.fpr* to see these filenames.

14.6 Running the Simulation

You are now ready to run the analysis. The analysis module of *FESWMS* is called *FST2DH* and it can be launched from inside *SMS*. To launch the *FST2DH* program:

1. Select *FESWMS* | *Run FST2DH*.

This command performs two basic tasks. These are:

1. Performing a model check to detect missed components. If no problems are detected, this step produces no visible effects. If the model is missing a required component (for example, if no boundary conditions exist), or if there is an error in the simulation (such as an invalid mesh domain), a list of problems is posted for the user.
2. Running the simulation. Once the check is complete, *SMS* launches the *FST2DH* executable. The location of the executable is stored as a model preference. The progress of the model is displayed in the *Model Wrapper*.

For this simulation, *FST2DH* should finish quickly. The *Model Wrapper* dialog waits for the user to acknowledge the completion of the model run. By default, it will then load the solution file when the *Exit* button is clicked. (If you are running in *Demo Mode*, the solution *stmary_ready.flo* is found in the *tutorials/tut14/output* directory and can be opened with the *File|Open* command.)

With the solution loaded, you are ready to evaluate the results. To do this:

1. Open the *Display Options*  dialog.
2. Under the *2D Mesh* tab, check the *Contours* and *Vectors* options.
3. Under the *Contour Options* tab, select *Color Fill* as the *Contour Method*.
4. Under the *Vectors* tab, select *Scale length to magnitude* as the option for *Shaft Length*. Click *OK* to close the *Display Options* dialog.

The *FST2DH* solutions for velocity magnitude, water depth and water surface elevation can be viewed by selecting the desired data set in the *Project Explorer*.

14.7 Conclusion

This concludes the Basic FESWMS Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File* | *Exit*. If prompted to confirm, click the *Yes* button.

FEWSMW Analysis with Weirs

15.1 Introduction

This lesson will teach you how to prepare an *FLO2DH* simulation which includes the use of weirs. You will be using the file *suecreek.sms* as shown in Figure 15-1. This mesh has been created and renumbered. To open the mesh data:

1. Select *File | Open*.
2. Highlight the file *suecreek.sms* in the *tutorials/tut15* directory and click the *Open* button. If a mesh is already open, you will be warned that the previous mesh will be deleted. If this happens, click the *OK* button at the prompt.

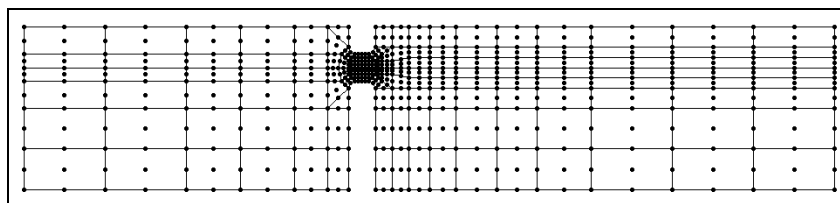


Figure 15-1. The *suecreek.fil* geometry.

15.2 Defining Material Properties

Each element of the mesh is assigned a material ID. The material ID tells *FLO2DH* which material properties should be assigned to the element. There are four different

material types in this mesh, but the material properties have not been defined. When SMS opens a mesh with undefined materials, the materials are assigned default properties. See the *FESWMS* documentation for a definition of individual material parameters. To change the material values:

1. Select *FESWMS | Material Properties*. A graphical image in the upper right corner of the *FESWMS Material Properties* dialog shows the Manning's n value as a function of water depth.
2. Highlight *material_01*, and enter the following values:
 - On the *Roughness Parameters* tab enter 0.035 for both $n1$ and $n2$
 - On the *Turbulence Parameters* tab enter 20.0 for Vo and 0.6 for $Cu1$
3. Select *material_02* and enter the same values that were entered for *material_01*.
4. For *material_03* and *material_04*, enter the same values as above, except that $n1$ and $n2$ are both 0.055.
5. Click the *OK* button to accept these changes and close the *FESWMS Material Properties* dialog.

You have just assigned values for the four materials in this mesh. Notice that there are only two distinct material regions because materials 1 and 2 have the same values, as do materials 3 and 4.

Optional: The materials can be displayed by opening the *Mesh Display Options* dialog and turning on the *Materials* option. You may turn on the display of materials, but if you do so, be sure to turn them off before continuing with this tutorial.

15.3 Assigning Boundary Conditions

Boundary conditions such as flow and head define how water enters and leaves the finite element network. Without proper boundary conditions, instability of the model and inaccuracy of the solution will result.

A steady state model such as this can only have constant boundary conditions. The flow and head boundary conditions will be defined at nodestrings on opposite sides of the model as shown in Figure 15-2. To create the two boundary nodestrings:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. Click on the lower node of the left boundary.
3. Hold the *SHIFT* key and double-click on the upper node of the left boundary. This creates the nodestring all the way across the left boundary of the mesh.

If you did not hold the *SHIFT* key, it would not be a valid boundary nodestring because it would not include all nodes across the boundary.

4. Repeat this procedure to create a nodestring across the right boundary.

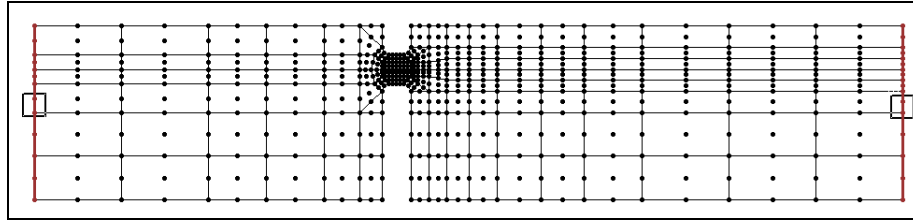


Figure 15-2. Location of the boundary condition nodestrings.

Boundary conditions can now be assigned to the nodestrings. To assign the flow to the left boundary:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring, as shown in Figure 15-2.
2. Select the nodestring on the left boundary by clicking inside its icon.
3. Select *FESWMS | Assign BC*.
4. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Flow* option and assign a value of 9000 (cfs). Be sure that the *Normal* option is selected.
5. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a flow nodestring and its color changes. An arrow appears at the center of the nodestring to indicate the flow direction and the flow value is shown next to the arrow (see Figure 15-3).

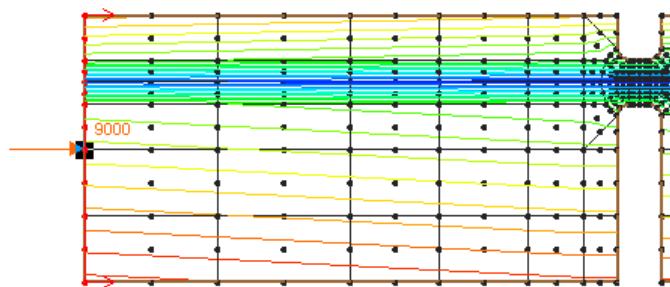


Figure 15-3 The inflow boundary condition.

To assign the head to the right boundary:

1. Select the right nodestring.

2. Select *FESWMS | Assign BC*.
3. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Water surface elevation* option and assign a value of 812.9 (feet). Be sure that the *Essential* option is selected.
4. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a head nodestring and its color changes. A head symbol appears at the center of the nodestring and the head value is shown next to the symbol (see Figure 15-4).

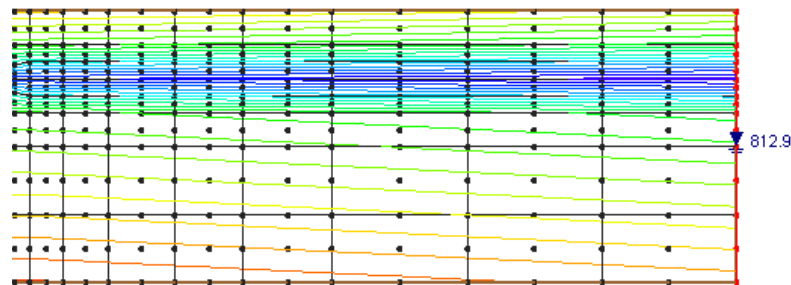


Figure 15-4 The outflow boundary condition.

15.4 Creating Weirs

With *FESWMS*, flow control structures such as weirs, piers, culverts, and drop inlets are easily added to the mesh. Weirs, culverts, and drop inlets are created between pairs of nodes. Wide structures can be created between strings of node pairs. For this model, a weir will be defined along five node pairs across the abutment at the bottom middle of the mesh, as shown in Figure 15-5.

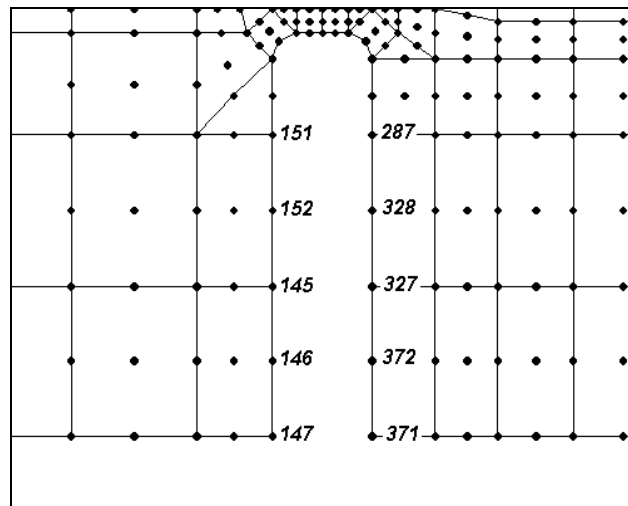


Figure 15-5 Area where weirs will be added.

This image highlights the nodes across which weir segments will be created. To see these nodes more clearly:

1. Zoom  in to the area shown in the above image.
2. Open the *Display Options*  dialog and turn on the *Node Numbers* option then click *OK*.

The five node pairs for the weir are: 151<->287, 152<->328, 145<->327, 146<->372, and 147<->371. A set of weir segments can be created across pairs of adjacent nodes by using *nodestrings*. To create the weir segments:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. Hold *SHIFT* and create one *nodestring* from node 151 to 147 and a second from node 287 to 371.
3. Choose the *Select Nodestrings*  tool and select both *nodestrings*.
4. Select *FESWMS | Weir*. Make sure the *Upstream* *nodestring* is that across nodes 151 to 147.
5. Enter the following values:
 - *Weir type* should be “Paved roadway”.
 - 825 for the *Zc – Crest Elevation*.
6. Click the *OK* button to close the dialog.

A set of five consecutive weir segments spanning the two bottom elements has just been defined. Together, these segments define a 300-foot-long broad crested weir with a crest elevation of 825 feet, and discharge coefficient of 0.544. SMS displays tick marks to show the break between weir segments. There is a specific formula for determining the length of each pair of nodes. Each midside node has 2/3 of the element width in its crest length while each corner node has 1/6 of each element width that is involved in the weir. See the *SMS Help* for more information on weirs and other flow control structures.

After creating the weir, reset the display to the way it was before starting the weir creation, as shown in Figure 15-6. To do this:

1. Turn off the display of node numbers using the *Display Options* dialog.
2. Click the *Frame*  macro in the *Toolbox* to frame the image.

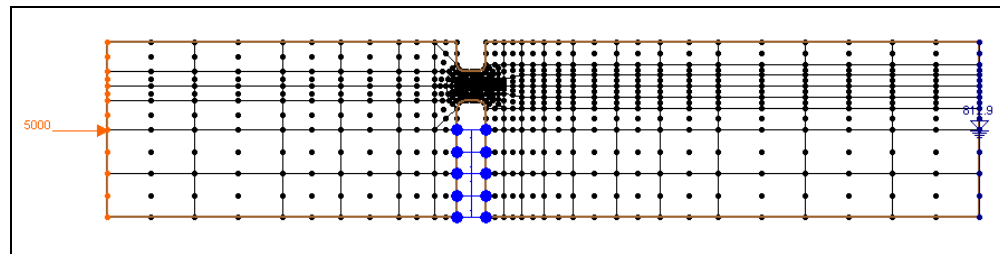


Figure 15-6. The bridge geometry with weirs.

15.5 Saving The Data

With *FESWMS* software, data is saved in multiple files. The file names are specified in the *FESWMS Model Control* dialog. To set up the *FESWMS* file options:

1. Select *FESWMS | Model Control*.
2. Click the *Parameters* button and turn on *Element drying / wetting*.
3. Click the *OK* button to close both dialogs.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *File | Save As*.
2. Make sure the *Save as type* is *Project Files* and enter the name *suecreek2*.
3. Click the *Save* button.

15.6 Using FLO2DH

You are now ready to run an analysis. The analysis module of *FESWMS* is called *FLO2DH*. It uses either a previous solution or a default value as initial conditions to compute the solution. The default conditions correspond to still water. If we tried to start from still water, this simulation will crash (feel free to try it using *FESWMS | Run FLO2DH*). However, SMS can start with lower flow and increase it to the desired rate using the steering module. To run *FLO2DH* through the steering module:

1. Select *Data | Steering Module*.
2. Select the *Flow* toggle and enter values of 50.0 and 0.1 as the *Init. Cond.* and *Min. Step*.
3. Click the *Run FLO2DH* button.

A window will open to run the *suecreek* model through *FLO2DH*. It will run the model with 50 percent of the flow, and then use that solution as an initial condition to run with 100 percent of the flow. Then you will get a notice that the steering process is complete. Click the *Exit* button to leave the dialog.

The final solution is stored in a file called *steer2.flo*. This file contains the velocity and water surface elevation for each node in the mesh. The solution can be read into *SMS* using the *File|Open* command. (The solution is also in the provided *output* directory.)

15.7 Editing Weir Data

In the previous solution, there is no flow over the weir. The water surface at the weir is much lower than the weir crest elevation so no overtopping occurred. This was purposely done to add model stability. With an initial solution, the weir's crest elevation can be lowered to allow overtopping. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox* and select the two weir nodestrings.
2. Choose *FESWMS | Weir*.
3. Change the *Crest Elevation* to *812.5 feet* and click the *OK* button.

15.7.1 Using The Hot Start file

SMS needs to tell *FLO2DH* to use the previous solution file as an input *hotstart*, or *initial condition*, file. To do this:

1. Select *FESWMS | Model Control*.
2. In the *FESWMS Control* dialog, turn on the *INI file* option.
3. Click the *File Browser*  button to the right of this option. Find and select the file *steer2.flo* that was created when *FLO2DH* ran. If you were not able to run the model, you can use the solution file that is provided in the *output* directory.
4. Click the *OK* button in both dialogs.

15.7.2 Computing a New Solution File Using a Hotstart File

To run the new simulation:

1. Select *File | Save As*, change the “Save at type:” to *FESWMS Simulation* and save the simulation as *suecreek3.fpr*.
2. Choose *FESWMS | Run Flo2dh* and click *OK*.

After flo2dh finishes, the solution file can be opened into *SMS* for post processing. See Lesson 6 for post processing operations.

15.8 Checking Flow Over Weirs

When flo2dh runs, it saves information for each weir segment. To view this:

1. Select *File | View Data File* and open the file *suecreek3.prt*.
2. Choose to open the file using Notepad.
3. From within notepad, select *Edit | Find* and search for the text *WEIR REPORT*.
4. Hold the F3 key to keep finding weir reports until a message appears that no the text cannot be found. This will be the last weir report in the file.

The final weir report should look like the following:

```
*** WEIR REPORT ***
=====
Weir id  Node 1  Node 2  Flow
         no.  WS elev  Energy  no.  WS elev  Energy  Flow rate Submerge
         no.  (ft)   (ft)    no.  (ft)   (ft)   (ft^3/sec) factor
-----
weir      147  814.048  814.068  371  813.157  813.182  144.757  1.000
weir      146  814.057  814.073  372  813.149  813.171  583.932  1.000
weir      145  814.065  814.078  327  813.141  813.159  294.425  1.000
weir      152  814.065  814.077  328  813.127  813.143  588.711  1.000
weir      151  814.065  814.086  287  813.114  813.122  147.143  1.000
=====
```

The second-to-last column shows the flowrate over each weir segment. To see which segment corresponds with which pair of nodes, turn on node numbers inside *SMS*.

15.9 Conclusion

This concludes the **Error! Reference source not found.** tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

Incremental Loading (Steering Module)

16.1 Introduction

This lesson will teach you how to use the steering module to perform incremental loading. Incremental loading has also been referred to as spinning down or revising the model. The process involves repetitively running the model with boundary conditions getting closer to the desired values. The steering module is used to automate the process. The methodology applies to both *FESWMS* and *RMA2*. *FESWMS* is used in this situation. The geometry has already been created and renumbered. To open the file:

1. Select *File | Open*.
2. Open the file “Capitol_Reef.fpr” from the *tutorial\tut16* directory. If you have mesh data open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The geometry data will open, as shown in [Figure 16-1](#).

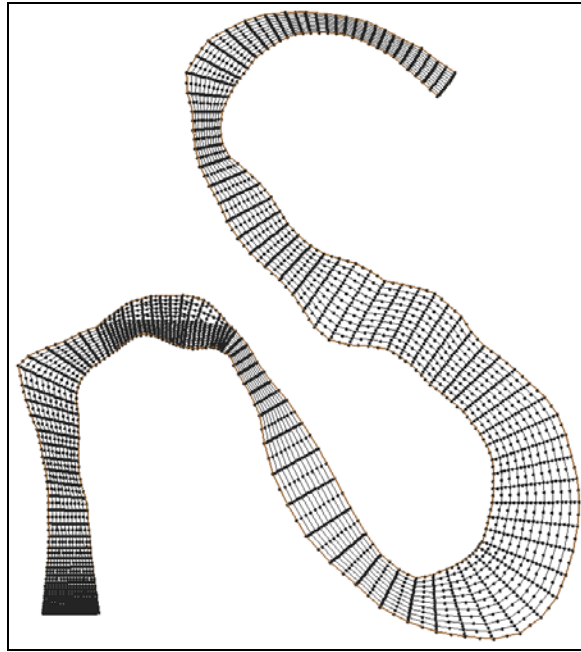


Figure 16-1. The mesh contained in the file *Capitol_Reef.fpr*.

16.2 Specifying Model Units

Before continuing, make sure that the units are English. To do this:

1. Select *Edit | Current Coordinates*.
2. Make sure the *Horizontal System* is set to *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*.
3. Click *OK* to exit the dialog.

16.3 Defining Model Parameters

Several model control parameters must be assigned to define the state of the model. These model parameters include items such as the input and output files, how to handle wetting and drying, the convergence parameters, and the number of iterations to be performed by *FESWMS*. Additional information on these parameters is found in the *FESWMS Help* and the *FESWMS* documentation. To define the model parameters:

- Select *FESWMS | Model Control*.

This opens the *FESWMS Control* dialog, in which the model parameters are controlled. This dialog contains various items such as simulation titles that describe what is being modeled. To set a title:

1. Click in the *Network Stamp* edit field and type, “Capitol Reef National Park.”
2. Click in the *BC Descriptor* edit field and type, “50 Year Flood.”

Input and output files may also be managed in the main dialog including the option to use ini files which are used to hotstart the model. This is desirable in complicated networks that require several steps to arrive to a solution. However we will use a different method that automates this process in this tutorial.

Before continuing, make sure that the solution type is set to steady state.

16.3.1 Iterations

The *Iterations* button contains options for defining the relaxation factor, number of iterations, and time steps in a dynamic model. The relaxation factor will not be explained at this point and this is a steady state model, so the only applicable parameter is the number of iterations. To set this value:

1. Click the *Iterations* button.
2. Enter 25 as the number of iterations.
3. Click OK.

16.3.2 Parameters

The *Parameters* button is used to set the general parameters of the model. We need to set the initial water surface elevation, the convergence parameters, and the wetting/drying parameters. To set these parameters:

1. Click the *Parameters* button.
2. Set the *Water-surface elevation* to 5070 ft.
3. Set the *Unit flow convergence* to 0.05 and the *Water depth convergence* to 0.005.

To accept all of the above values:

- Click the *OK* button twice to close the *FESWMS Parameters* dialog and the *FESWMS Control* dialog.

16.4 Defining Boundary Conditions

For this tutorial, flowrate and water surface elevation will be defined along nodestrings at the *open boundaries* of the mesh. An open boundary is a boundary where water is allowed to enter or exit. Generally for *FESWMS*, a flowrate is specified across inflow boundaries and water surface elevation is specified across outflow boundaries. Other available boundary conditions are rating curves and reflecting boundaries.

This model has one inflow boundary and one outflow boundary so two nodestrings must be created. These boundaries are highlighted in [Figure 16-2](#).

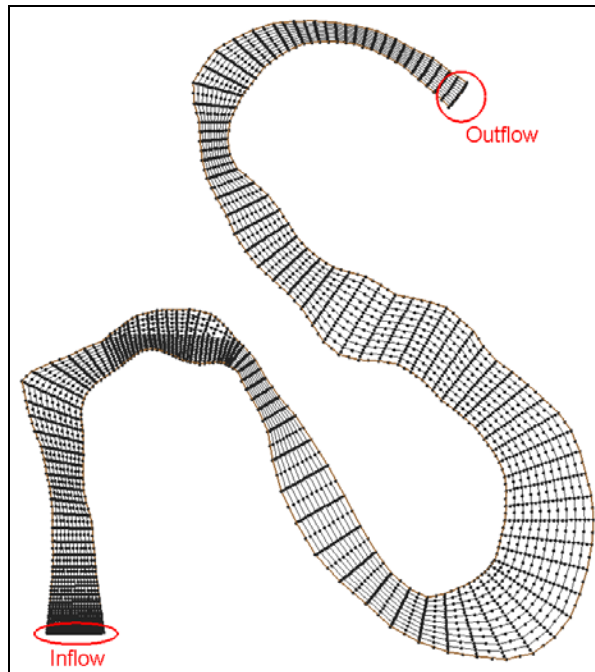


Figure 16-2. Position of the boundary nodestrings in the mesh.

16.4.1 Creating Nodestrings

Nodestrings should be created from right to left when looking downstream and the first nodestring should be that which spans across the whole river section. In this case both nodestrings span the entire river section so it does not matter which nodestring is created first. To create the outflow nodestring:

1. Choose the *Create Nodestrings* tool  from the *Toolbox*.
2. Start the nodestring by clicking on the lower node at the outflow boundary.
3. Hold the *SHIFT* key and double-click on the upper node at the outflow boundary to create and end the nodestring.

The inflow nodestring can be created now. To do this:

- Create a nodestring across the inflow boundary. Make sure you create it right to left when looking downstream.

16.4.2 Defining Flow Boundary Conditions

To assign the flow condition:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring.
2. Select the bottom inflow nodestring by clicking on the icon.
3. Select *FESWMS | Assign BC*.
4. Make sure the *Boundary Type* is set to *Specified Flow / WSE* and select the *Flow* checkbox.
5. Assign a constant *Flowrate* of 6,550 (cfs).
6. Click the *OK* button to assign the boundary condition.

This defines the bottom nodestring to be an inflow boundary condition.

16.4.3 Defining Head Boundary Conditions

A water surface elevation (head) boundary condition will be assigned to the outflow boundary nodestring. To assign this boundary condition:

1. Select the outflow nodestring.
2. Select *FESWMS | Assign BC*.
3. Make sure the *Boundary Type* is set to *Specified Flow / WSE* and select the *Water surface elevation* checkbox.
4. Assign a constant value of 5,070 (feet).
5. Click the *OK* button to assign the boundary condition.

16.5 Defining Material Properties

Each element in the mesh is assigned a material type ID. This particular geometry has five material types. To see each of these materials:

1. Select *Display | Display Options* or click the *Display Options*  macro.
2. In the *2D Mesh* tab, turn on the *Materials* option.
3. Turn off the *Nodes* and *Elements* options.
4. Click the *OK* button to close the *Display Options* dialog.

The display should look something like [Figure 16-3](#). Most of the model is made of brush floodplain and the channel, but there are a few elements with other material types.

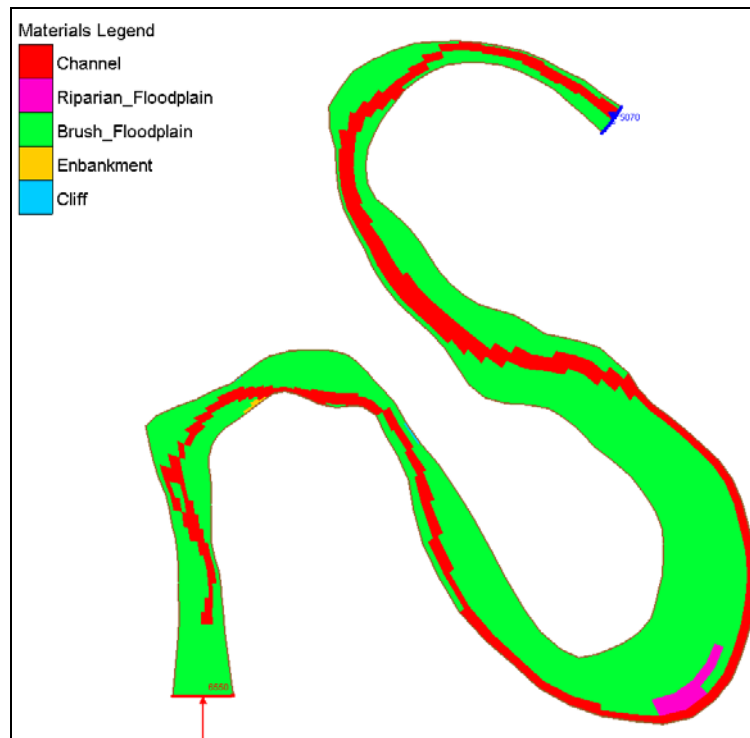


Figure 16-3. The display of materials.

Before continuing, turn off the material display. To do this:

- Open the *Display Options* dialog. Turn off the *Materials* option and turn the *Elements* options back on.

The materials were created with default parameters that must be changed for this particular simulation. The material properties define how water flows through the element. To edit the material parameters:

1. Select *FESWMS | Material Properties*.
2. In the *FESWMS Materials Properties* dialog, highlight *Brush Floodplain*.

3. Under the *Roughness* tab, set both *n1* and *n2* to 0.05.
4. Under the *Turbulence* tab, make sure the eddy viscosity (*Vo*) is set to 20 for this material.
5. Set the roughness for Channel to 0.03, Cliff to 0.05, Embankment to 0.04, and Riparian Floodplain to 0.1. Make sure you set both *n1* and *n2*. The viscosity for each material should already be set to 20.
6. Click the *OK* button to close the *FESWMS Material Properties* dialog.

The eddy viscosity and roughness parameters have now been defined for this model.

16.6 Saving the Simulation

SMS can launch the *FST2DH* numerical model. However, the model will read the data from files. Therefore, the simulation must be saved prior to running. If you haven't done this, you will be prompted to do so. To save the simulation:

1. Select *File | Save As*.
2. Change the *Save as type* to *FESWMS Simulation (*.fpr;*.fil)* and enter "CR_Sim" as the *File name*.
3. Click the *Save* button to save the simulation.

16.7 Running the Model

At this point, you are ready to try running *FST2DH*. To do this, choose *FESWMS | Run FST2DH*. Before *SMS* launches the model, a quick check is done on the data to make sure everything is valid. This model check will bring up the dialog shown in [Figure 16-4](#) if any anomalies are detected. For this model, three warnings should be detected.

The first warning says that the elements might dry out, so the wet/dry flag should be turned on. To do this:

1. Select the *Cancel* button to leave the *Model Checker* without running *FST2DH*.
2. Select *FESWMS | Model Control*.
3. Click the *Parameters* button.
4. Set the *Default storativity depth* to 0.1 ft.

5. Turn on *Element drying / wetting*, and set the *Depth tolerance for drying* to 0.1 ft.
6. Click the *OK* button twice to close the *FESWMS Parameters* dialog and the *FESWMS Control* dialog.
7. Resave the simulation, and select the *FESWMS | Run* command again to assure that the first error has gone away.
8. Then click the *Run Model* button to launch the *Model Check* again.

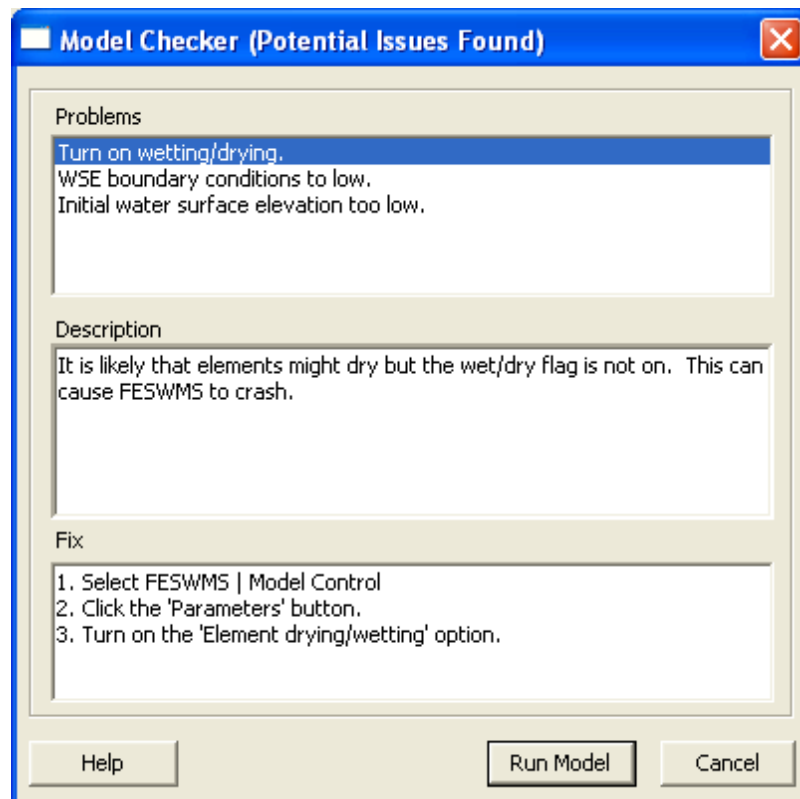


Figure 16-4. Warning in FESWMS data.

The remaining warnings say that the initial water surface elevation and the WSE boundary condition for the simulation are too low and will leave portions of the domain dry. This can lead to instabilities and cause FST2DH to crash. One option is to raise the initial water surface elevation, but if it is much higher than the outflow boundary elevation, instabilities can develop at those locations. If the simulation is not stable at the WSE boundary condition, it may be best to use incremental loading as will be demonstrated later. For now we will ignore these warnings. Because there is still a warning message, FST2DH might not converge. However, before attempting to fix this, the simulation will be tried.

- Click on *Run Model* to launch FST2DH.

FST2DH will run iteration 1 and diverge. At the bottom of the *FESWMS Output* window is the text “***Run ended abnormally because of 1 errors and 0 warnings!” as highlighted in [Figure 16-5](#). This is how *FST2DH* declares that it has not successfully converged.

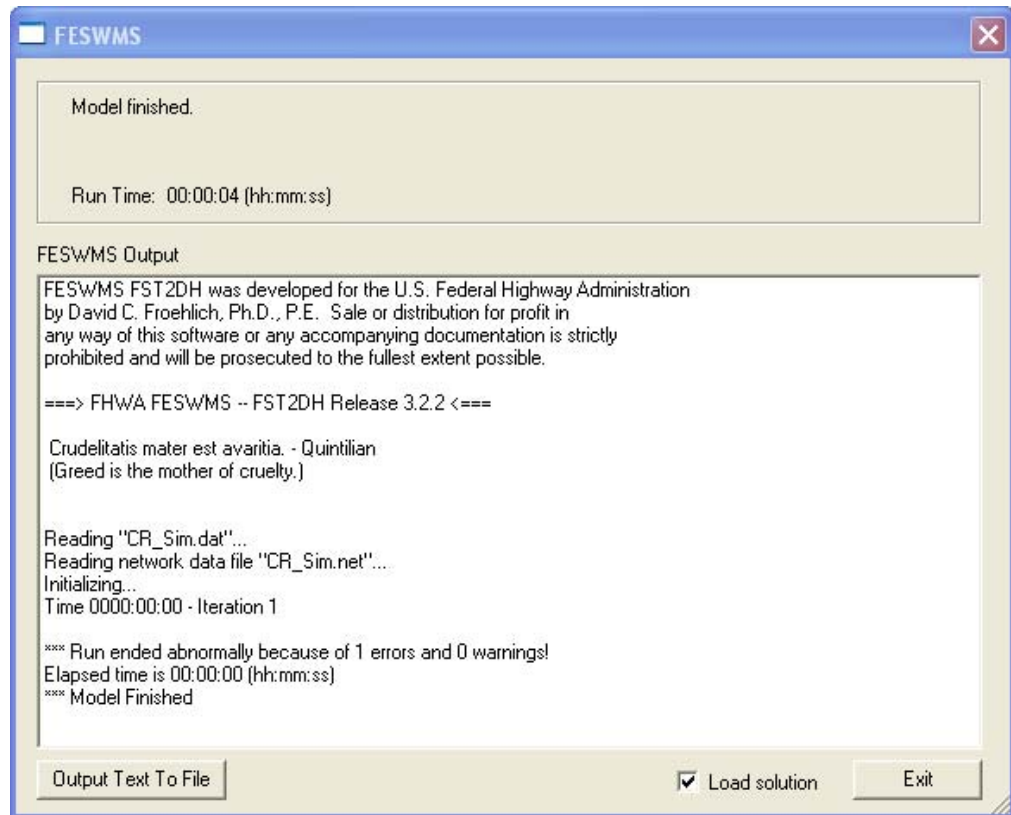


Figure 16-5. Output from running *CR_Sim.fpr*.

- Uncheck the *Load Solution* option and click the *Exit* button to close the *FESWMS Output* window.

Various things can contribute to a model not converging. In this case, *SMS* had given an error message that the initial water surface was low. The low water surface elevation for this simulation does not allow *FST2DH* to converge from the coldstart simulation. To illustrate why this occurs, we will compare the boundary condition with the bathymetry. To do this:

1. Select *File | Get Info* or click the *Get Info*  macro.
2. In the *Mesh Info* tab, look at the *Max Z value*, highlighted in [Figure 16-6](#).

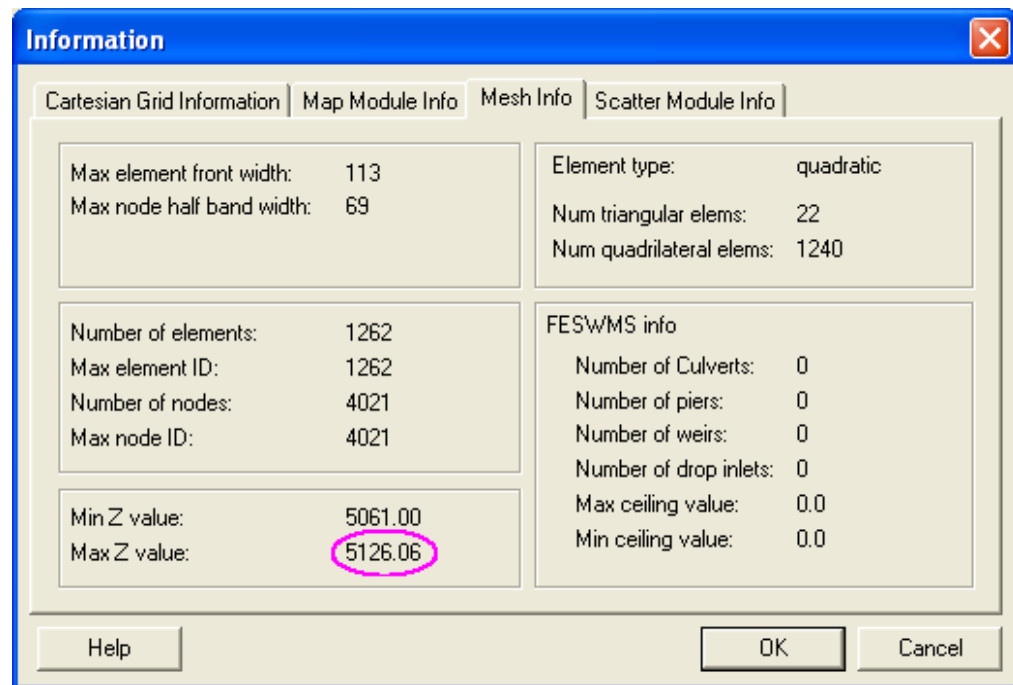


Figure 16-6. Mesh Information dialog.

You can clearly see that the maximum bathymetry elevation is well above our boundary condition of 5,070 ft. This means not all of the nodes are wet with our initial water surface elevation. FST2DH requires that all nodes be wet for the initial condition, or the model will not run. This is why FST2DH diverged after only running the first iteration.

- Close the *Information* dialog by clicking OK.

16.8 Using the Steering Module

It is possible to set the boundary water surface elevation high enough to wet all nodes. The model could then be run and an output solution file could be created. This solution file could be used to hotstart the model with a lower boundary water surface elevation. This process could then be repeated until the boundary water surface elevation is at the desired level, but that would require an enormous amount of user input and time, especially for this model. Fortunately, this process can be automated with no user-interaction by using the steering module. To run the steering module:

1. Select Data | Steering Module.
2. In the *Steering Module* dialog select *Water Surface Elevation* for the *FST2DH Spindown* method as shown in [Figure 16-7](#).
3. Click *Run FST2DH*.

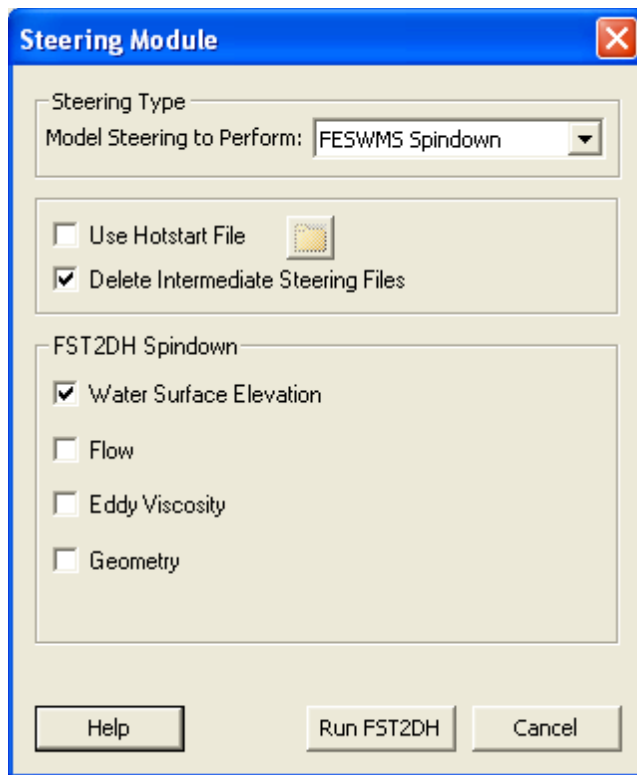


Figure 16-7. Steering Module dialog.

The FESWMS Spin-down dialog shown in [Figure 16-8](#) will keep you updated on the progress of spinning down the model.

The top window in the dialog explains the WSE convergence of the current run. Each iteration is shown as a green point, and you can tell if the run is converging or diverging depending on if the points are moving towards or away from 0 head change. The iteration being performed is shown just above this plot.

The bottom window shows the overall spin down of the model. The green points represent successful runs, and the red Xs represent failed runs. When this plot reaches 100% spun down, then the model is finished. This percent is shown just above this plot.

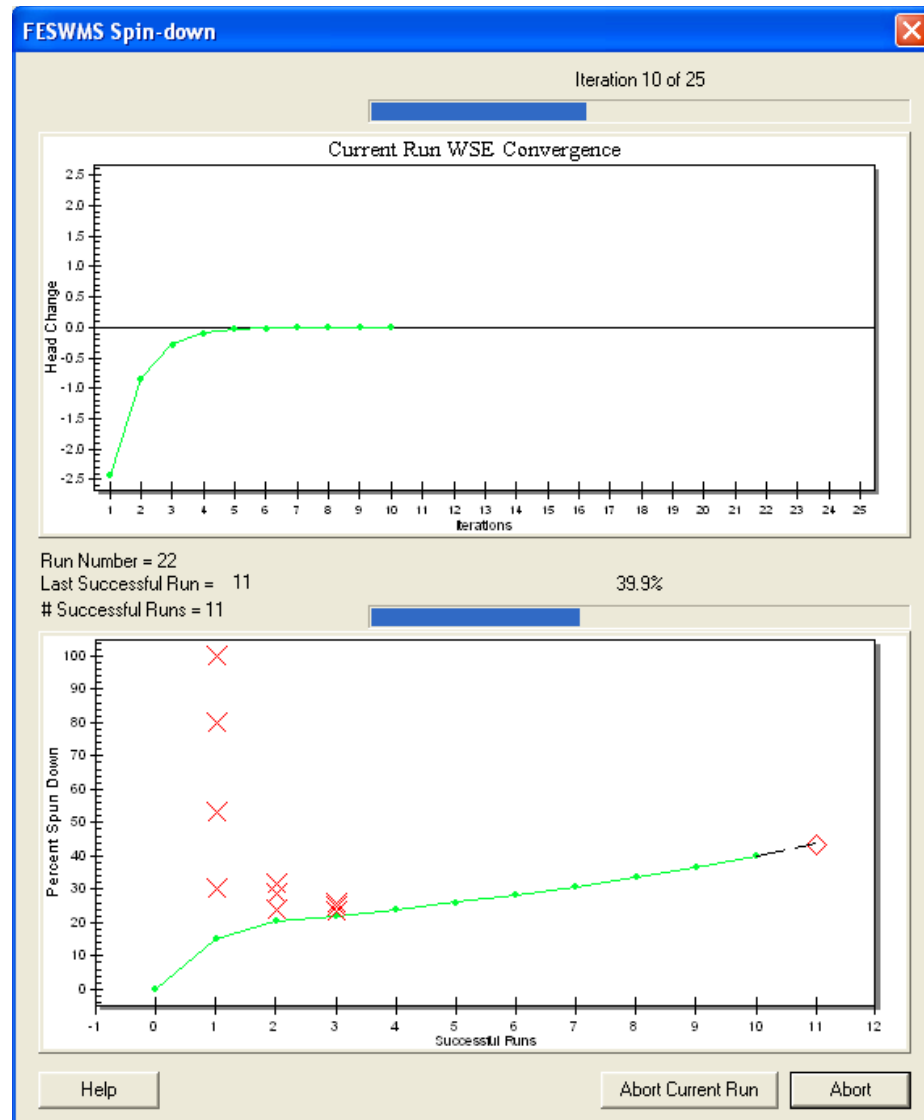


Figure 16-8. FESWMS Spin-down dialog.

16.9 Opening the Solution

The entire spindown process takes a few minutes depending on the speed of your computer. When it has finished, a window will open telling you that the “Steering process has terminated – See status file for details”. This status file is named “SteeringStatus.txt” and gives a summary of the steering process. A final solution file has also been created. To view the solution of the spindown:

1. Click OK and click OK again in the *FESWMS Spin-down* dialog.
2. Select File | Open and open the solution file named “CR_Sim.flo”.

16.10 Conclusion

This concludes the [Incremental Loading \(Steering Module\)](#) tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

ADCIRC Analysis

17.1 Introduction

This lesson will teach you how to prepare a mesh for analysis and run a solution for *ADCIRC*. It will cover preparation of the necessary input files for the *ADCIRC* circulation model and visualization of the output. You will start by reading in a coastline file and then a SHOALS file.

The data used for this tutorial is from Shinnecock Bay off of Long Island in New York. All files used for this tutorial are located in the *tutorial\tut17* directory.

17.2 Reading in a Coastline File

For this tutorial, you will first read in a coastline file, which has already been set up for you. This sample coastline will form the boundary for your mesh.

To open the coastline file:

1. Select File | *Open*.
2. Select the file *shin.cst* and click the *Open* button.
3. Because you will be working with *ADCIRC* meshes, select the *ADCIRC* button as the coverage you want to create.

Coastline files include lists of two-dimensional polylines that may be closed or open. The open polylines are converted to *Feature Arcs* and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands.

17.2.1 Defining the Domain

We need to define the region to be modeled. To do this:

1. Make sure you are in the *Map*  module.
2. Choose the *Select Feature Arc*  tool from the *Toolbox* and click on the coastline arc to select it.
3. Choose *Feature Objects* | *Define Domain*.
4. Select the *Semi-circular* option and push OK.

A semi-circular arc is created to define the region.

17.2.2 Assigning Boundary Types

Boundary types for the initial mesh generation are specified in the *Map* module. Boundary types are prescribed by setting attributes to Feature Arcs. To set the boundary types:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Double click the arc representing the ocean boundary, shown in Figure 17-1.
3. In the *ADCIRC Arc Atts* dialog, assign this arc to be of type *Ocean*.
4. Click the *OK* button to close the dialog.
5. Repeat this process for the coastline arc, but make sure the boundary type is set to *Mainland*, as shown in Figure 17-1.

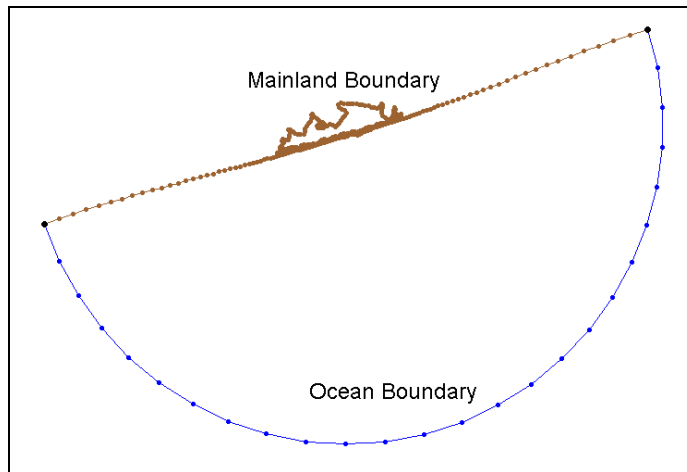


Figure 17-1 Feature Arcs after boundary types have been assigned.

17.3 Editing the Coastline File

Now that the coastline file has been read in, several modifications must be made to the data before the SHOALS file is read in.

17.3.1 Coordinate Conversions

The current coordinates of the coastline file are in a latitude/longitude geographic coordinate system. However, in order to work through this particular tutorial, we need to convert the coordinates to UTM. This will aid you in making calculations and will help preserve the accuracy of the mesh you will build later in the lesson. To convert the coordinates:

1. Choose *Edit | Coordinate Conversions*.
2. In the *Coordinate Conversion* dialog that appears, click the *Current Coordinate System...* button to define the coordinate system the data is currently in.
3. In the *Current Coordinates* dialog, change the coordinates for the *Horizontal System* to *Geographic NAD 27 (US)*.
4. Since the bathymetric data you will be reading in is in meters, set the *Units* for the *Vertical System* to *Meters*.
5. Click the *OK* button to close the *Current Coordinates* dialog.

6. In the *Coordinate Conversion* dialog, in the *Coordinates* section of the dialog, set the *Horizontal System* to *UTM NAD 27 (US)*. Make sure the *UTM Zone* is set to *Zone 18 78W to 72W*.
7. Change the *Units* for both the *Horizontal* and *Vertical Systems* to *Meters*.
8. Click the *OK* button to exit the dialog.

The coastline data has now been converted from Geographic coordinates to UTM coordinates. The coastline should now look skewed from the original coastline.

17.4 Reading in a SHOALS File

You will now read in a SHOALS file, *shin.pts*, which contains data at various locations along the coastline and throughout the region you are modeling.

1. Choose *File | Open*.
2. Select the file *shin.pts*.

The *File Import Wizard* dialog will open, allowing you to specify how the data will be read into SMS. For *Step 1* of the dialog, the first line in the *File preview* box is the file header. The next line shows the name of each respective column of data. In this case, the file has three data columns. The first column is the *X Coordinate*, the second column is the *Y Coordinate*, and the third column is the *depth/bathymetry*.

- Click the *Next >* button to move on to *Step 2* of the *File Import Wizard*.

The second step of the *File Import Wizard* allows you to change other specifications as you read in the SHOALS file.

- Change the data in the 3rd column to be elevation (z) dataset by changing the combo-box at the top of the column to z.
- Click the *Finish* button.

Figure 17-2 shows the plot of the points read in from the *shin.pts* file.

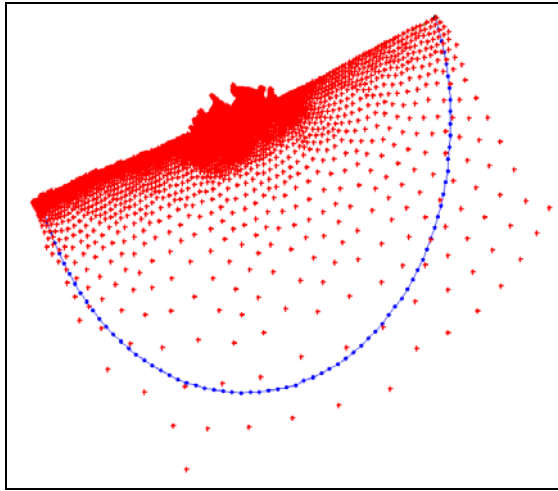


Figure 17-2 Display of shin.pts.

17.5 Shallow Wavelength Functions

The next step before you build your finite-element mesh is to create several functions for creating the finite element mesh. For this tutorial, the mesh will be generated according to the wavelength at each node. Large elements will be created in regions of long wavelengths. Conversely, smaller elements are needed closer to shore to correctly model the smaller wavelengths.

To create this shallow wavelength function from the bathymetric data:

1. In the Scatter  module, select *Data | Create Data Sets*.
2. In the *Create Data Sets* dialog, select the *All Off* button to turn off all of the functions.
3. Turn on the *Coastal* function option.
4. Turn on the option for creating a *Shallow Wavelength/Celerity* function. Leave the period at 20 sec.
5. Click the *OK* button to create the functions and close the dialog.

Two functions are created: celerity and wavelength at each node using shallow water wavelength equation. The celerity is calculated as:

- $Celerity = (Gravity * Nodal\ Elevation)^{0.5}$.

The wavelength is calculated as:

- $Wavelength = Period * Celerity$.

17.6 Creating Size Functions

Now that you have created the shallow wavelength function, you must make a few more conversions before you are ready to create your mesh. A size function is a multiple or variation of an already existing function. If you were to generate your mesh using the original wavelength function alone, you would get a decent mesh to work with, but we want a mesh whose density radiates out from a point in the inlet. This allows you to get more accurate results in the inlet where we are most concerned with the outcome of the *ADCIRC* run. Therefore, you now need to create a size function that works with the type of mesh we want. The final size function is found by using a variation of the original wavelength function. The final size function you will use was found through trial and error to give a nicely formed mesh. For this case, you will create several separate functions and then combine them into one final size function to use when meshing.

17.6.1 Finding the Central Point for the Mesh

Since the mesh will be generated in a radial fashion, the distance from a central point must be found. The first step is to locate the central point and then use the *Data Calculator* to compute the distances of all points from this center point. To do this:

1. Still in the *Scatterpoint*  module, zoom in to the area of the inlet as shown in Figure 17-3 using the *Zoom*  tool in the *Toolbox*.
2. Click on one of scatter points (Figure 17-4) in the middle of the inlet using the *Select Scatterpoints*  tool. Make note of this point's *X* and *Y* coordinates in the *Edit Window* at the top of the screen.
3. Frame the data by clicking the *Frame*  tool in the *Toolbox*.

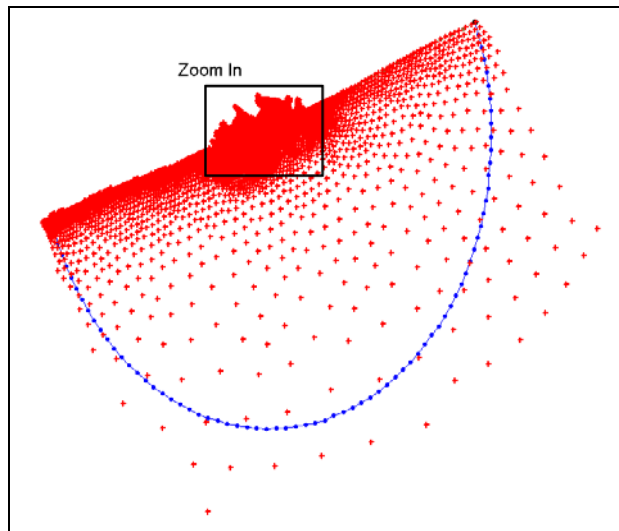


Figure 17-3 Inlet location to zoom in on.

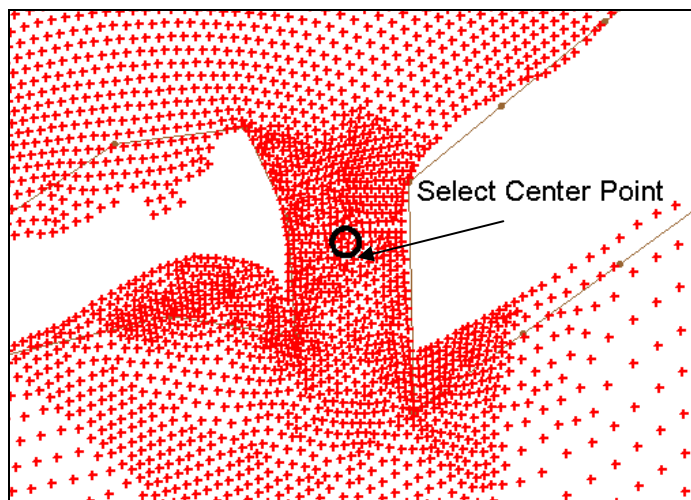


Figure 17-4 Choose a center point.

For now, turn off the scatterpoint display. However, you may turn it back on at any time during the tutorial if you so desire. To turn off the visibility of the *shin.pts* data:

1. Choose the *Display Options*  macro from the *Toolbox*.
2. In the *Display Options* dialog, under the *Scatter* tab, turn off the *Visible* option.
3. Click the *OK* button to close the dialog.

You are now ready to proceed. SMS has a powerful tool, called the *Data Calculator*, for computing new data sets by performing operations on scalar values and existing data sets. The *Data Calculator* will be used to create the size function.

17.6.2 Distance Function

For consistency, we will use the (x,y) location of (712768.675, 4523969.712) as the center scatterpoint for our mesh.

1. Select *Data | Data Calculator*.
2. Click the *sqrt(x)* button.
3. In the *Expression* field, replace “??” using the keyboard so the expression looks like:

$$\text{sqrt}((d - 712768)^2 + (e - 4523950)^2)$$

This expression takes the *x* and *y* locations of each scatterpoint, which correspond to the “d” and “e” data sets respectively, and computes its distance to the point designated as the mesh center.

4. In the *Result* field, enter the name of “distance” for the data set and click the *Compute* button.

17.6.3 Initial Size Function

1. Clear the *Expression* field (delete all characters in the edit field).
2. Highlight the “Shallow_Wavelength” data set and click the *Add to Expression* button. You now should see the letter “b” in the *Expression* field at the bottom.
3. In the *Expression* field, make the equation look like “b*7”.
4. Enter the name “size” for this data set in the *Result* area and click the *Compute* button. This creates a function of 7 times the wavelength.

17.6.4 Scale Function

The last separate function before computing the final size function will be a scale factor out from the center point. It will take on the following format:

$$\text{scale} = (\text{distance}/\text{max distance})^{0.5}.$$

This scale function will range between 0 and 1, 0 being at the center point and 1 at the farthest point from the center of the mesh. This will allow the mesh to radiate out in density from the middle of the inlet. Taking the square root of the scale factor forces the elements to grow larger more quickly as one moves away from the center. To compute this function:

1. Highlight the “distance” function in the *Data Sets* window and click the *Info...* button. Notice that the *Maximum value* is 65627.258.
2. Click the *OK* button to return to the *Data Calculator* dialog.
3. Clear the equation in the *Expression* field and enter “ $\sqrt{f / 65627.258}$ ”. This assumes that *f* is the distance function.
4. Enter the name “scale” and click the *Compute* button.

17.6.5 Final Size Function

You are now ready to create the final size function that your mesh will be based on.

1. Clear the equation from the *Expression* field.
2. Click the *max(x,y)* button.
3. Replace “??,??” so the equation reads “ $\max(50, (g*h))$ ”. This will multiply the scale factor by the size. The minimum size of the elements will be 50 meters to prevent infinitely small elements from being created around the mesh center.
4. Enter the name “finalsize” in the *Result* field and click the *Compute* button.
5. Click *Done* to exit the *Data Calculator* dialog.

The data calculator gives you many options for building the size function. The size function created in this tutorial was created through several steps. This was done to show the many possibilities that exist for defining the size function, and ultimately for defining the finite element mesh. Other options that could be used for this or other meshes include:

- Use the wavelength multiplied by a scale factor (without using distance).
- Don’t take the square root of the scale factor for a denser mesh.
- Use a value other than 50 meters as the minimum size for a denser mesh in the channel, etc.

17.7 Creating Polygons

A *polygon* is defined by a closed loop of Feature Arcs and can consist of a single Feature Arc or multiple Feature Arcs, as long as a closed loop is formed. For initial mesh generation, polygons are a means for defining the mesh domain.

17.7.1 Building Polygons

To create polygons from the arcs on the screen:

1. Switch to the *Map*  module.
2. Make sure that no arcs are currently selected.
3. Select *Feature Objects* | *Build Polygons*.
4. Now a polygon has been created out of all the arcs, that can be used for the mesh.

17.7.2 Polygon Attributes

Next each polygon must be selected and have all the proper attributes set.

1. Choose the *Select Feature Polygon*  tool from the *Toolbox* and click inside the polygon.
2. Select *Feature Objects* | *Attributes*. (Double-clicking inside the polygon will perform this same step.) The *Polygon Attributes* dialog will open.

17.7.3 Assigning the Meshing Type

1. Select *Scalar Paving Density* as the *Mesh Type*.
2. Click the *Scatter Options...* button below the *Mesh Type*.
3. In the *Interpolation* dialog, in the dataset tree under *Dataset to interpolate from*, highlight the *finalsize* function.
4. In the *Extrapolation* section, set the *Single Value* to 50.
5. Make sure the *Truncate values* option is turned on and set the *Min* to 50 and the *Max* to 5000.
6. Click the *OK* button to return to the *Polygon Attributes* dialog.

17.7.4 Assigning the Bathymetry Type

Next, the bathymetry type is selected. In this case the imported bathymetry is in the form of a scatter set.

1. Select *Scatter set* as the *Bathymetry Type*.

2. Click the *Scatter Options...* button below the *Bathymetry Type* option.
3. In the *Interpolation* dialog, make sure that *elevation* is the highlighted function under *Data Set*, leave the *Single Value* at 0.000, and make sure the *Truncate values* option is turned off.
4. Click the *OK* button.

17.7.5 Assigning the Polygon Type

1. Change the *Polygon/Type Material* to *Ocean*.
2. Click the *OK* button to close the *Polygon Attributes* dialog.

17.8 Creating the Mesh

Once the polygon attributes are set, the mesh can be generated automatically based on the options that were selected. To generate the mesh:

- Select *Feature Objects* | *Map* -> *2D Mesh*.

Depending on the speed of your computer, this process could take anywhere from several seconds to a few minutes.

17.8.1 Mesh Display Options

After SMS has completed generation of the mesh, you should be able to view the bathymetry, nodes, and elements. To set the display:

1. Switch to the *Mesh*  module.
2. Select *Display* | *Display Options...* or select the  macro from the *Toolbox*.
3. Turn both the *Nodes* and *Contours* off and the *Elements* on under the *2D Mesh* tab.
4. Click the *OK* button to close the *Mesh Display Options* dialog.

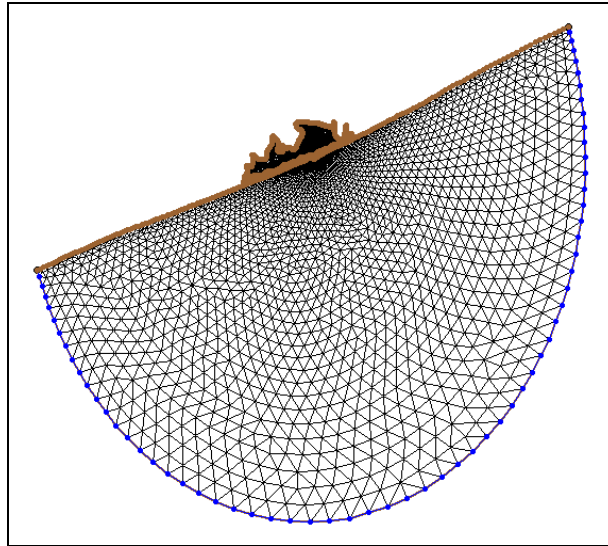


Figure 17-5 View of elements after automatic mesh generation.

Figure 17-5 shows the final mesh. Notice how the elements are smaller closer to the coast and within the inlet. Once the mesh has been created and refined, final preparations must be done in order to run *ADCIRC*. These items are renumbering of the mesh nodes and saving the grid.

17.8.2 Minimizing Mesh Bandwidth

Before running *ADCIRC*, the mesh nodes must be renumbered to minimize the bandwidth of the mesh. This allows the *ADCIRC* model to run efficiently. To do this:

1. Select the *Select Nodestring*  tool from the *Toolbox* and select the nodestring along the ocean boundary.
2. Choose *Nodestrings | Renumber*.
3. Make sure the *Bandwidth* option is selected and click the *OK* button.

The nodes have now been renumbered for the entire mesh starting with those along the ocean boundary.

17.9 Building the *ADCIRC* Control File

The control file specifies values corresponding to different parameters for *ADCIRC* runs. These parameters include specifications for tidal forcing, selection of terms to include, hot start options, model timing, numerical settings, and output control. In order for *ADCIRC* to run properly, the mesh must be converted to Latitude/Longitude coordinates.

17.9.1 Converting Back to Lat/Lon

The model control expects the coordinates to be in latitude/longitude. The initial conversion was made to UTM coordinates for the meshing. The size function was calculated in meters, so the mesh could not be created while the coordinates were in degrees without performing more conversions (i.e. degrees <-> meters). To convert back to Geographic coordinates:

1. Select *Edit | Coordinate Conversions*.
2. In the *Convert to* section, switch the *Horizontal System* to *Geographic NAD 27 (US)*.
3. Make sure the *Units* for the *Vertical System* are in *Meters*.
4. Click the *OK* button.

17.9.2 Main Model Control Screen

To set up the model control for *ADCIRC*:

1. Select *ADCIRC | Model Control*.
2. Turn on the following options under *Terms* in the center of the dialog: *Time Derivative Terms On*, *Advective Terms On*, *Finite Amplitude Terms On*, and *Wetting/Drying*.
3. Click the *Options...* button beside the *Wetting/Drying* option.
4. Make sure the following values are entered in the *Wetting/Drying Parameters* dialog:

<i>Minimum Water Depth</i>	0.05
<i>Minimum # of Dry Timesteps</i>	12
<i>Number of Rewetting Timesteps</i>	12
<i>Minimum Velocity for Wetting</i>	0.02
5. Click *OK* to return to the *Model Control* dialog.
6. Locate the *Model Center* by clicking the *Find Center* button (bottom right).
7. Enter a value of 3.0 for the *Lateral Viscosity*.

17.9.3 Time Control

Next, values for the *Time Control* must be set. To set these values:

1. Click the *Time Control* button (top right).

2. Set the following values:

<i>Time Step:</i>	4.0 seconds
<i>Run Time:</i>	0.1 days (2.4 hours)

ADCIRC will generate two global output files, water-surface elevation and velocity. To set the time for the two files:

Select the *Constituent* and scroll down until *Global Elevation* is highlighted.

1. Enter the following values:
2. *Start Day* 0.0
3. *Output Every* 10 *Minutes*
4. *End Day* 0.1
5. Change the *Constituent* to *Global Velocity* and enter the same values.
6. Click the *OK* button to return to the *Model Control* dialog.

17.9.4 Tidal Forces

For this run of *ADCIRC*, tidal forcing will be used. To define the tidal constituents that *ADCIRC* will apply at the ocean boundaries:

17.9.4.1 Tidal Potential Constituents

1. Click the *Tidal Forces* button from the main *Model Control* dialog.
2. To tell *ADCIRC* to run with tides, change the *Tidal Potential* to *On*.
3. Click the *New* button under *Tidal Potential Constituents*.
4. In the *New Constituent* dialog, make sure the *LeProvost* constituent database is selected. For this tutorial, you will be using the M2, N2, S2, O1, and K1 constituents.
5. Set the *Starting Day* as 0.00 hours on February 1, 2000 (0.00 hour, 1 day, 2 month, 2000 year). This is the date from which the tides will start.
6. Click the *M2* constituent from the *Constituents* box on the right and click the *OK* button to return to the *Tidal Functions* dialog.
7. Repeat steps 3 through 7 for the N2, S2, O1, and K1 constituents.

17.9.4.2 Tidal Forcing Frequencies Constituents

Now, we have to tell *ADCIRC* to use these same constituents as forcing constituents on the open boundary of our finite-element mesh.

1. Press the *Copy Potential Constituents* button. If a prompt appears that SMS cannot find *m2.legi*, push *OK* and find *m2.legi* in the file browser.

SMS takes each constituent and extracts the values it needs from the *LeProvost* constituent database, placing it into the *Tidal Forcing Frequencies Constituents* box on the right.

2. Click the *OK* button twice to exit the *ADCIRC Model Control* dialog.

17.9.5 Saving The Mesh and Control Files

To save the mesh and control files:

1. Select *File | Save Project*.
2. Enter the name *shinfinal.spr* and click the *Save* button.

17.10 Running *ADCIRC*

You are now ready to run *ADCIRC*. Presently, *ADCIRC* uses a specific naming convention for its input and output files. Therefore, before *ADCIRC* can start, the basic input files must be present in the working directory, which *SMS* does automatically. *SMS* makes a copy of the active mesh file and names it *fort.14*, then makes a copy of the model control information file and names it *fort.15*. The *ADCIRC* executable also needs to be located in the directory where the files are located.

To run *ADCIRC*:

1. Move or copy the *ADCIRC* executable into the *tutorial\tut17* directory.
2. Select *ADCIRC | Run ADCIRC*.
3. If the name of the *ADCIRC* executable does not appear, click the folder icon , locate the *ADCIRC* executable, and click *OK*.

ADCIRC will run in a DOS window for 2160 timesteps. This will take just over 8 minutes on a Pentium 3, at 800 Mhz

Once the *ADCIRC* run has completed, there will be several new files created. *SMS* copied the *shinfinal.grd* file (the mesh file saved when the project file was saved) to

fort.14 and *shinfinal.ctl* file to *fort.15*, the filenames needed by *ADCIRC*. *ADCIRC* created the *fort.63* (global elevation) and the *fort.64* (global velocity) files. There are a couple of other files that hold basic output information, but we will only focus on the elevation and velocity files for the remainder of this tutorial.

17.11 Importing *ADCIRC* Global Output Files

Each output file from *ADCIRC* is imported into *SMS* as a “Dataset.” There are two types of datasets, scalar and vector. The global elevation file is an example of a scalar dataset, while the global velocity file is a vector dataset. You will first import the global elevation file. To do this:

1. Select *Data | Data Browser*.
2. In the *Data Browser* dialog, click the *Import...* button.
3. Change the *Files of type:* to *ADCIRC Unit 63 (*.63;*.sol)*.
4. Select *fort.63* and click the *Open* button.
5. In the *Dataset File Options* dialog, select the *Add to Solution Set* option and click the *OK* button.

SMS now reads the file and adds “water surface elevation (63)” in the *Scalar Data Sets* windows under the *Generic Dataset* solution.

6. Repeat items 2 through 4, this time selecting the file *fort.64*. Changing the *Files of type:* to *ADCIRC Unit 64 (*.64;*.sol)*.

Notice that *SMS* adds “velocity (64)” to the *Vector Data Sets* window. Additionally, *SMS* has created a new scalar dataset named “magnitude (64)” that contains the magnitude of the current velocity and has placed it into the *Scalar Data Sets* window.

7. Click *Done* to exit the *Data Browser* dialog box.

17.12 Viewing *ADCIRC* Output

Once both *ADCIRC* output files have been imported, the user must decide on how to view the data. Look at the *Edit Window* at the top of the screen. This shows the presently selected solution and data sets. Notice that the *Scalar* dataset is currently *magnitude (64)*.

- Change the *Scalar* dataset to “water surface elevation (63)”.

17.12.1 Scalar Dataset Options

A good way to view the output is to edit the contour display options. To change the contour properties:

1. Select the *Display Options*  macro in the *Toolbox*.
2. Click the *All off* button to turn off current display options.
3. Turn on the *Contours*, and *Mesh boundary*.
4. Under the *Contours* tab, change the *Contour Method* to *Color Fill*.
5. For the *Number of contours*, enter 25.
6. Turn on the *Specify a range* option.
7. Enter -0.60 for the *Minimum value* and 0.10 for the *Maximum value*.
8. Make sure both the *Fill below* and *Fill above* boxes are checked.
9. Click *OK* time to exit the dialog box, and *SMS* will redraw the screen similar to Figure 17-6. Note (It will look about the same for the *Time step* 480.)

To view the data at different time steps, change the value of the presently displayed *Time Step* in the *Edit Window* at the top of the screen. The *Time Step* value is in seconds from the start of the *ADCIRC* run.

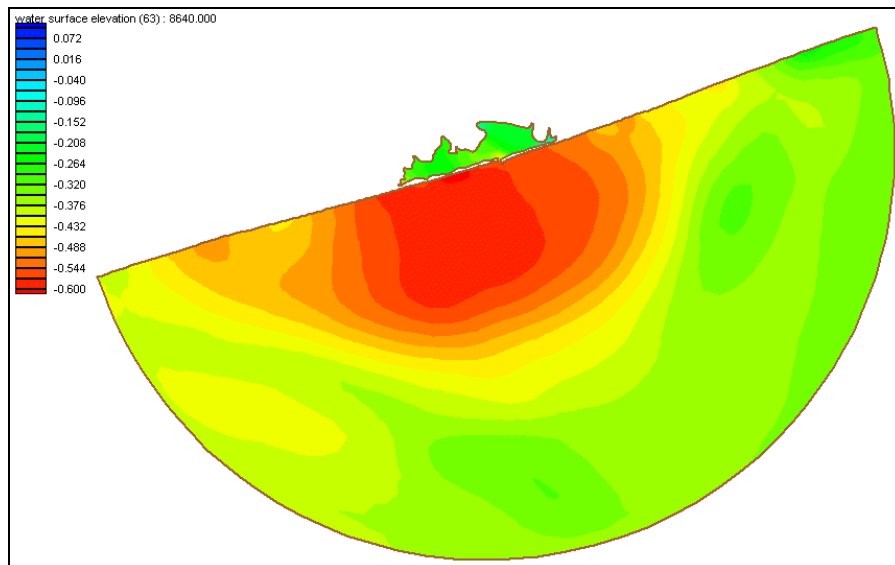


Figure 17-6 ADCIRC output from the "fort.63" file.

17.12.2 Vector Dataset Options

You can display velocity vectors several different ways. We will first view them displayed at each node, and then on a normalized grid.

17.12.2.1 Vectors at Each Node

1. Using the *Zoom* tool , zoom in on the mesh so only the bay area is visible.
2. Select the *Display Options*  macro in the *Toolbox*.
3. In the *Display Options* dialog under the *2D Mesh* tab, turn on the *Vectors* toggle.
4. Under the *Vector* tab, under the *Shaft Length* select *Define min and max length*.
5. Change the *Min length* to 10 *pix* and click the *OK* button.
6. Click the *OK* button to exit.

The screen should now look similar to that shown in Figure 17-7. You can now visualize the flow at each node through Shinnecock Bay at this particular time step.

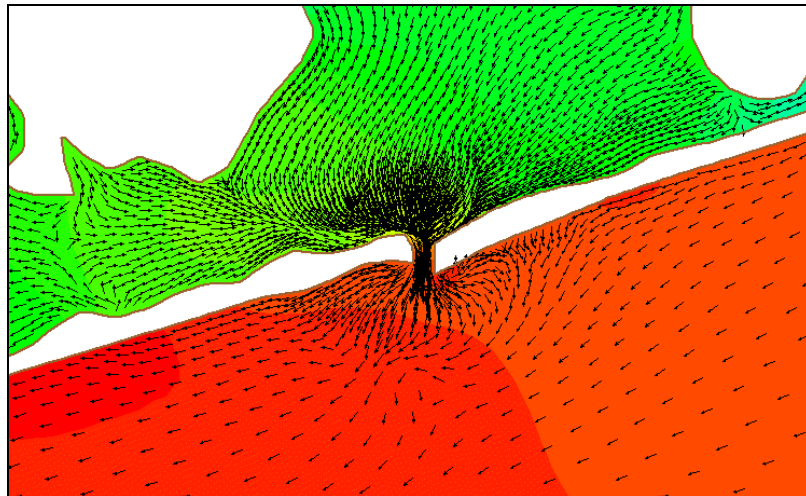


Figure 17-7 View of velocity vectors at each node.

17.12.2.2 Vectors on a Normalized Grid

1. Since the Velocity Vectors are already active, click the *Vector Options* tool  on the *Toolbox*.
2. Change the *Arrow Placement* to the *Display vectors on a Grid* option.
3. For both the *x pix* and *y pix*, enter a value of 15 and click the *OK* button.

An example of equally spaced vectors is shown in Figure 17-8. This method of displaying vectors is useful when displaying areas with both coarse and refined areas.

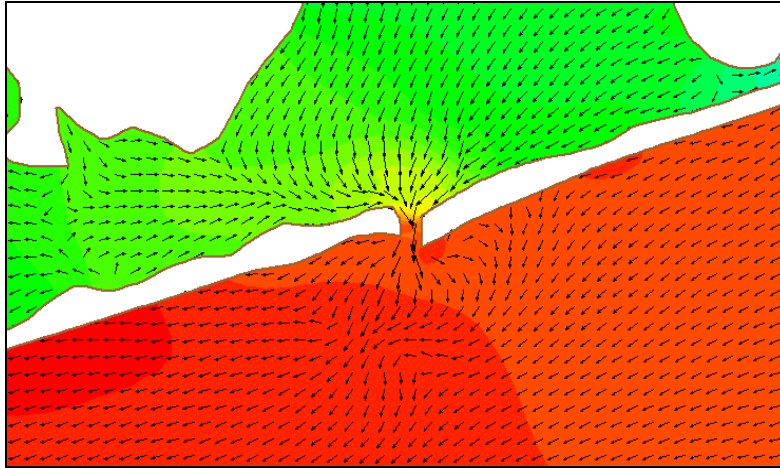


Figure 17-8 View of velocity vectors on a normalized grid.

17.13 Film Loop Visualization

Once the water-surface elevation contours and the velocity vectors are set, animations can be generated and saved. SMS enables the user to generate and save animations by using the Film Loop. To create a film loop of the ADCIRC analysis:

1. Select *Data | Film Loop*.
2. In the *Film Loop* dialog, click the *Next>* button.
3. Turn on both the *Scalar* and *Vector data sets* under the *Data Options* and click the *Next>* button, then the *Finish* button. (Note this will take several minutes to run.)

SMS now starts the film loop, adding one frame at a time. Once the last frame has been added to the loop, you can view the animation. To do so:

1. Click the *Play*  button.
2. You can have the animation start over again when it reaches the last time step by pressing the  button or you can have it go in reverse when it reaches the last time step by pressing the  button.

You may continue to experiment with the film loop features if you desire. Click the *Close* button when finished. You may save the film loop if you want to.

17.14 Conclusion

This concludes the *ADCIRC* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

STWAVE Analysis

18.1 Introduction

This workshop gives a brief introduction to the *STWAVE* modules. Data from the Shinnecock Inlet, Long Island, New York, has been set up as an example. This example will use the mesh generated in the previous tutorial for *ADCIRC*. An *STWAVE* grid will be created over a small section of the *ADCIRC* mesh.

18.2 Converting ADCIRC to Scatter

18.2.1 Reading in the ADCIRC files

First, open the mesh and solution files generated in lesson 13. The files are also supplied in the *tutorial\tut18* directory. To open the files:

1. Select *File | Open...* and select *shinfinal.grd*. Push *Open* to read in the mesh file.
2. Open *fort.64* in the same manner as step 1.

The mesh will be drawn on the screen after the first step. To see the velocity vectors from the solution file:

1. Select *Display | Display Options* or the  macro in the *Toolbox*.

2. Turn on the *Vectors* toggle under the *2D Mesh* tab and push *OK*.

18.2.2 Converting to Scatter

The mesh file was read into the *Mesh* module . We need to convert the data into scattered data points (which are accessed in the *Scatter* module ). Scattered data points are used for interpolating between meshes and Cartesian grids. To convert the data to scatter points.

1. Still in the *Mesh* module , go to *Data | Mesh -> Scatterpoint*, leave the name as *scatter*, and push *OK*.
2. We won't be using the mesh anymore so turn the display off by going to *Display | Display Options* , push the *All off* button under the *2D Mesh* tab, and push *OK*.

Next, we can delete the unnecessary scatter points for speed and memory purposes:

1. Switch to the *Scatter* module . Zoom  in on the dense part as shown in Figure 18-1. Zoom by dragging a box around a region or single clicking. Hold the *Shift* key while clicking to zoom out.
2. Select the *Select Scatterpoints*  tool and choose *Edit | Select With Poly*. This allows you to click on the screen and draw out a polygon. Any points inside the polygon will be selected. Select the points around the box shown in Figure 18-1 with about a 30% margin. Double-click on the screen to close the polygon and select the points.
3. Go to *Scatter | Split Scatter Set*. Set the name to *Grid* when prompted and push *OK*.
4. Push the *Frame*  button to show both scatter sets.
5. We no longer need the larger scatter set so go to *Scatter | Delete Scatter Set*. Select *scatter* and click the *Delete* button near the bottom, and push *Done*.

Note: This is one method of reducing a scatter set. We could have also selected the points that we didn't want and deleted them.

We now have a scatter set that covers the area of interest.

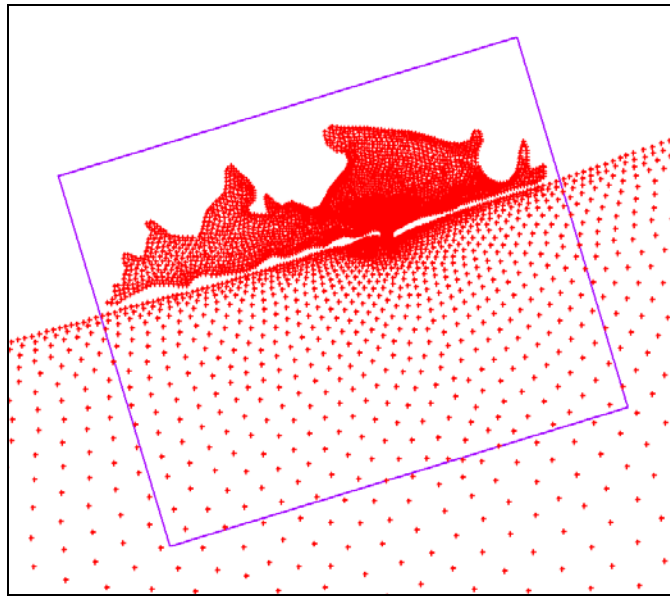


Figure 18-1. Zoomed in view of the scatter data set.

18.3 Creating the Cartesian Grid

We will now create a Cartesian grid for running *STWAVE*. The grid frame is created in the *Map* module . The *Map* module contains tools for creating GIS objects such as points, arcs, and polygons. It is also used for creating a frame, which will be filled in by a Cartesian grid.

18.3.1 Creating the Cartesian Grid Frame

To create the grid frame:

1. Switch to the *Map* module .
2. Go to *Feature Objects | Coverages...* and switch the *Coverage Type* to *STWAVE*. Push *OK*.
3. Select the Create Grid Frame tool .
4. Click out three corners of the grid to create the grid frame. Switch to the Select Grid Frame tool . Drag and resize the grid frame by dragging the corners or edges until the grid frame fits over the desired area. Dragging a corner resizes the frame. Dragging an edge moves the entire frame.
5. Rotate the grid by dragging the circle at the bottom right corner of the frame. **IMPORTANT:** Rotate the frame until the circle is at the top right of the

screen. The origin of the grid frame is now the frame corner at the bottom right of the screen.

6. Push *OK* when you are done positioning the frame to appear as Figure 18-1.
7. *Frame*  the data on the screen.

18.3.2 Creating the Land and Ocean Polygons

Before filling the interior of the grid frame with cells, we need to specify which cells will be ocean cells and which cells will be land cells.

1. Go back to the *Mesh*  module and choose *Data | Mesh -> Map*.
2. Select the *Mesh Boundaries -> Polygons* option and deselect the *Create New Coverage* option. This creates map polygons around the boundaries of the mesh in the active coverage. Push *OK*.
3. Switch back to the *Map*  module.
4. Select the *Create Arcs*  tool and create arcs surrounding the top part of the grid frame (outside of the polygons created in Step 1). Click to begin an arc and click to create vertices. Double click to end the arc. Connect the arcs with vertices of the ocean polygon as shown in Figure 18-2. Make sure you click on a vertex to close the loop.
5. Go to *Feature Objects | Build Polygons*. This creates polygons from closed loops of arcs.
6. Select the *Select Polygon*  tool. Double-click in the new polygon that you created (on the top) and select *Land* and push *OK*. The default for the big polygon on the bottom is *Ocean*.
7. Click somewhere on the screen outside of the polygons to deselect the polygon.

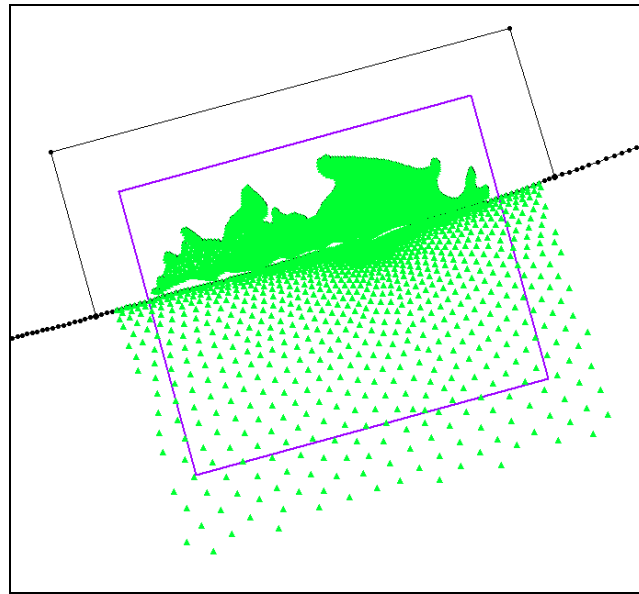


Figure 18-2. Arc created around border of grid frame.

18.3.3 Mapping to the Grid

We are now ready to fill the interior of the grid. While the grid is filling, the depth and current values will be interpolated from the scatter set and mapped to each cell. To do this:

1. Go to *Feature Objects* | *Map->2D Grid*.
2. In the *Map -> 2D Grid* dialog, set the *Cell Options* to *Number of Columns* and set the value to 70.
3. Make sure that the *elevation* function will be used to interpolate depth values for each cell in the *Depth* section.
4. Turn the *Current* toggle on. Make sure the *Interpolated* option is selected and click the button to the right of it.
5. In the *Interpolated* dialog, change *Single Time Steps* to *Multiple Time Steps*.
6. Enter “240” in *Start time*, “900” in *Step size* and “2940” in *End Time*. This tells SMS to create current data at times of 240, 1140, 2040, and 2940 seconds (four sets of values around three steps of 15 minutes each).
7. Select *OK* to exit the *Interpolated* dialog.
8. Push *OK* to create the Cartesian grid.

Note on this particular grid: Because the width of the inlet and the spacing between the bay and the ocean are so small, you may need to experiment with several

different sized grids to produce one that works. This is best done by moving the grid frame around and re-sizing it. However, make sure that the grid frame bounds the area of interest. Do this until the grid most closely resembles the original mesh, with only one inlet into the bay.

Note on interpolation: When interpolating you can specify a single time step or multiple steps. Single times can come from any time in the data set or interpolated between two of these times. For multiple steps, you can specify to match the steps from the data set for a specified number of steps, or you can specify a new time step size and the number of steps. SMS will generate one set of values at the beginning time specified, and one at the end of each time step. Therefore, there will be “n+1” sets of currents if you specify “n” time steps.

A Cartesian grid has been created from the grid frame. To view the grid only:

1. Go to *Feature Objects | Delete* and say *OK* at the prompt.
2. Switch to the *Scatter* module  and select *Display | Display Options* .
3. Turn the *Visible* toggle next to *Grid* off and push *OK*.
4. Switch to the *Cartesian Grid* module  and select *Display | Display Options* .
5. Turn on the *Ocean Boundary* and push *OK*.

18.4 Editing the Grid and Running STWAVE

18.4.1 Generating Spectral Energy Distribution

We will now generate the Spectral Energy distribution.

1. Switch to the *Cartesian Grid* module  and choose *STWAVE | Spectral Energy*.
2. Click the *Create Grid* button to bring up the *Create Spectral Energy Grid* dialog.
3. In the *Frequency Distribution* section of the dialog, change the *Number* to 40 and click *OK* to create a new spectral energy grid.
4. The new spectral energy grid will appear in the *Spectral Manager* tree control in the *Spectral Energy* dialog.
5. Make sure the new grid is selected and click the *Generate Spectra* button.

6. In the *Generate Spectra* dialog, enter the following parameters into the spreadsheet.

Index	Angle	Hs	Tp	Gamma	nn	Depth
1	25.0	1	20	3.3	4	35

7. Push the *Generate* button. The new spectrum, labeled “1,” will appear below the grid in the *Spectral Manager* tree control. Select the spectrum. The contours show the energy distribution. Select cell corners to view/edit their energies.
8. Push *Done* to exit the *Spectral Energy* dialog.

18.4.2 Model Control

In the Model Control, *STWAVE* inputs can be set. We will change the Wind parameters:

1. Select *STWAVE | Model Control*.
2. Click the *Specify Spectral Parameters* button. This allows us to specify *STWAVE* specific parameters associated with the spectrum just created. Enter 0.0 in for *Wind Speed*, *Wind Direction*, and *Tidal Elevation*. Click *OK* to close the dialog.
3. Click the *Select Input Spectra* button. This allows us to specify which spectra the current *STWAVE* run will use. Check the toggle box next to the spectrum labeled “1.” Click *OK* to close the dialog.
4. Change the *Source Terms* under *Other Settings* to *Propagation Only*.
5. Push *OK* to exit the dialog.

18.4.3 Selecting Monitoring Stations

The final step is to select cells to act as monitoring stations:

1. Select the *Select Cell*  tool.
2. Select a cell in the inlet. The i,j location can be seen in the bottom of the screen in the status portion of the *Edit Window* when a cell is selected. (Click on the screen to select a cell.)
3. While the cell is still selected, choose *STWAVE | Assign Cell Attributes*. Turn the *Monitoring Station* toggle on and push *OK*.

4. Repeat steps 2 and 3 and select a cell in the ocean and one in the bay to be *Monitoring* cells. You can also choose an exact i, j or x, y location by going to *Data | Find Cell*.

18.4.4 Saving the Simulation

To save the simulation:

1. Select *File | Save As*, make sure the *Save as type* is set to *Project Files*, and enter the file name *shin.spr*.
2. Push the *Save* button.

18.4.5 Running STWAVE

To run *STWAVE*:

1. Select *STWAVE | Run STWAVE*.
2. If a message such as “stwave.exe – not found” is given, click the *File Browser* button  to manually find the *STWAVE* executable.
3. Click the *OK* button to launch *STWAVE*.

A dialog will come up showing the progress of the *STWAVE* run as well as the exact screen output from *STWAVE* if it was run from DOS.

18.5 Post Processing

The solution can be opened in SMS and several visualization options may be set.

18.5.1 Visualizing the STWAVE Solution

To see the solution results:

1. Open the solution files by selecting *File | Open*, open the following files *shin_CGrid1.wav*, and *shin_CGrid1.obs*. The naming convention for CGRID files when a project file is saved is the project file name followed by double underscores and finished with the name of the grid.
2. Select *Display | Display Options* . Under the *Cartesian Grid* tab turn the *Contours* and *Vectors* toggles on.
3. Under the *Contours* tab under the *Contour Method* select *Color fill*.

4. Under the *Vector* tab change the *Shaft Length* to *Define min and max length*.
5. Set the *Min length* to 25 and the *Max length* to 50.
6. Change the *Display vectors* under *Arrow Placement* to *on a grid*.
7. Push *OK* to exit the *Display Options* dialog.

18.5.2 Visualizing Bathymetry

The solution files are stored as an SMS solution set when they are opened. The depth is not part of the solution output from *STWAVE*; it is part of the “Generic Datasets” solution. To view the bathymetry:

- Change the Solution (in the *Edit Window* near the top of *SMS*) from “shin (*STWAVE*)” to “Generic Datasets”. Notice that the current *Scalar* function (again at the top) is changed to “Depth.”

18.5.3 Visualizing the Direction Field

The *STWAVE* solution files are stored in the new solution. To see the wave direction vectors:

1. Switch the *Solution* back to “shin (*STWAVE*)” in the *Edit Window*.
2. Change the *Vector* function to “wave.”

The vector arrows show the wave direction field.

18.5.4 Visualizing the Wave Height

To see the wave height:

1. Change the *Scalar* function to “Height.”

The contours show the wave height. Repeat step 1 for “Period” and “Direction.”

18.6 Conclusion

This concludes the *STWAVE* Analysis tutorial. If you wish to exit *SMS* at this point:

- Choose *File | Exit*.

CGWAVE Analysis

19.1 Introduction

This lesson will teach you how to prepare a mesh for analysis and run a solution for CGWAVE. You will start with the data file *indiana.xyz* which contains a set of points that contain depth data from which a mesh will be created. To open the data:

1. Select *File* | *Open*.
2. Select *indiana.xyz* in the *tutorial\tut19* directory and click the *Open* button.
3. The *File Import Wizard* dialog will appear. Click *Next* to proceed to Step 2 of the *File Import Wizard*.
4. Map the three columns of the file as X, Y, and Z, respectively.
5. Push *Finish* to open the file. (This wizard allows you to open data that may not have data in 3 columns of x, y, and z. Data in any number of columns in any order can be opened through the wizard).

A scatter set named Indiana will be created and will appear in the *Project Explorer*. This data is referenced to a UTM coordinate frame and is in meters. To give this information to SMS: select *Edit|Current Coordinates* and set the units to meters.

19.2 Creating a Wavelength Function

The first step in creating a mesh for *CGWAVE* is to create a wavelength function. The wavelength function is an intermediate step to creating a size function, which is explained in section 19.3. The *z* value of each point in the *indiana.xyz* data is actually a water depth value. The wavelength at each point is calculated from this depth value using a complicated equation. It is sufficient to say that a larger wavelength is calculated from a larger water depth value. To create the wavelength function:

1. Select the “Indiana” data set in the *Project Explorer* (this makes the scatter module active).
2. Select *Data | Create Data Sets*.
3. Turn off everything but the *Transition Wavelength/Celerity* option. (The *Coastal* option must be on to access the *Transition Wavelength/Celerity* option.)
4. Leave the function name as *Transition* and enter a *Period* of 20 seconds.
5. Click the *OK* button.

Two new data sets will be created, one named *Transition_Wavelength* and the other named *Transition_Celerity*. These can be seen in the *Project Explorer*.

19.3 Creating a Size Function

The size function is created from the wavelength function. The size function is the function that determines the element size that will be created by *SMS*. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller.

The wavelength function that was created in section 19.2 contains values that are twice as large as the desired size values. The wavelength function will be scaled by one half to create the size function. To do this:

1. Select *Data | Data Calculator*.
2. In the top left section of the *Data Calculator*, highlight the function named *Transition_Wavelength* and click the *Add to Expression* button. The letter that represents this function will appear in the *Expression* field.
3. Click the / (divide symbol) in the middle section of the *Data Calculator*.
4. After the divide symbol, enter the number 5 (five) using the keyboard. (Note: This “5” represents that we will generate approximates five elements per

wavelength. It is usually more appropriate to use a larger number of elements per wavelength (10). The smaller number is used here to allow faster execution of the model.)

5. In the *Result* field, enter the name *size* and then click the *Compute* button.
6. When the computation is completed, *size* will appear as a data set.
7. Click the *Done* button to exit the *Data Calculator*.

A new data set named *size* that was created based on the *Transition_Wavelength* data set should appear in the *Project Explorer*.

19.4 Defining the Domain

A domain represents the region that is offshore. In *CGWAVE*, the domain can be a circular, semi-circular, or rectangular region. In *SMS*, a *Feature Arc* is used to define the coastline. After the coastline is defined, *Feature Arcs* and *Feature Polygons* are used to define the domain region.

19.4.1 Creating the Coastline

SMS can automatically create a coastline at a specific elevation or water depth from a scattered data set. The active function of the active scattered data set will be used for this operation. You should currently have only one scattered data set. To make the elevation function active:

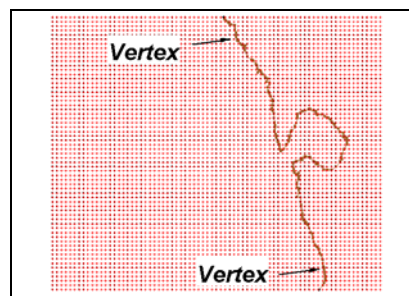
1. Select the elevation (Z) function in the *Project Explorer*. (This makes the elevation function current in the scatter module.)
2. Right click on the “default coverage” item and select “Type”. In the menu that appears change the type to “CGWAVE”.
3. Select the “default coverage” (left click) to make it the active object.
4. With the coverage type set and the active scattered data set defined, you are ready to create the coastline. Select *Feature Objects* | *Create Coastline*. You will be prompted for *Elevation* and *spacing* values. Enter the value of 1.0 and click the *OK* button..

The display will refresh with an arc representing the 1.0 water depth line, as shown in Figure 19-1a.

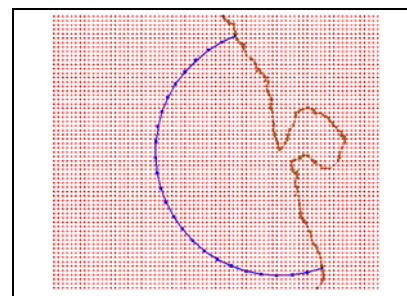
19.4.2 Creating the domain

SMS can create a domain from the coastline. This model will use a semi-circular domain that intersects with the coastline. To create the domain:

1. Choose the *Select Feature Vertex*  tool from the *Toolbox*.
2. Hold the *SHIFT* key and click on two vertices, one near each end of the coastline arc, as shown in Figure 19-1a.
3. Select *Feature Objects | Define Domain*. Select *Semi-circular*, and click *OK*. This creates a semicircular *Ocean* arc as shown in Figure 19-1b. If the domain is created on the wrong side of the coastline, it indicates that the coastline is oriented in the wrong direction. If this happens:
 - Choose the *Select Feature Arc*  tool from the *Toolbox*
 - Select the semi-circular arc and delete it.
 - Select the coastline arc and reverse its direction using the *Feature Objects | Reverse Arc Direction* command.
 - Repeat the command in step 3 to create the domain.



(a). The coastline feature arc.



(b). The domain feature arc.

Figure 19-1 The indiana scatterpoint and feature data.

Now that feature arcs define the domain, a feature polygon must be created from the feature arcs. To create the polygon:

- Select *Feature Objects | Build Polygons*.

After this command is executed, polygons are formed from any set of arcs that form a closed loop. The screen will not refresh when polygons are built, so it may appear that nothing happened even though polygons were created. For this example, there should now be a single polygon made from the semi-circular ocean arc and the part of the coastline arc with which it intersects.

19.5 Creating the Finite Element Mesh

There are various automatic mesh generation techniques that can be used to create elements inside a specified boundary. One of these is applied to each polygon, after which a finite element mesh can be generated. For this tutorial, there is only one polygon, which will be assigned the *Density mesh* type.

19.5.1 Setting-Up the Polygon

When using density meshing, SMS determines element sizes from a *size function* in a scattered data set. The size function to be used in this example was created back in section 19.3. To set up the feature polygon for density meshing:

1. Choose the *Select Feature Polygons*  tool from the *Toolbox*. With this tool selected, double-click inside the polygon that defines the domain.
2. In the *Polygon Attributes* dialog, change the *Mesh Type* to *Scalar Paving Density* and press the *Scatter Options* button.
3. In the bottom left of the *Interpolation* dialog, turn on the *Truncate values* option and set the *Min* and *Max* to 10 and 10000, respectively. This sets up a minimum and maximum size to be used when creating elements.
4. Select *size* as the *Scatter Set to Interpolate From*.
5. Click the *OK* button to get back to the *Interpolation* dialog. (If a warning appears about the extrapolation value, click *OK*.)
6. In the *Bathymetry Type* section, select *Scatter set*.
7. Select the *Scatter Options* option under *Bathymetry Type*, highlight the function named *elevation* as the *Scatter Set to Interpolate From* and make sure the *Truncate Values* option is turned off. As mesh nodes are created, their elevation value will be assigned from the original water depth values that were read from the xyz file.
8. Click the *OK* button to close both dialogs.

The polygon is now set up to generate finite elements inside the boundary. When more than one polygon exists, the meshing attributes need to be set up for each of the polygons.

19.5.2 Generating the Elements

Since there is only one polygon in this example, you are ready to have SMS generate the finite element mesh from the defined domain. To create the mesh:

- Select *Feature Objects* | *Map->2D Mesh*.

After a few moments, the mesh will be created to look something like the finite element mesh in Figure 19-2.

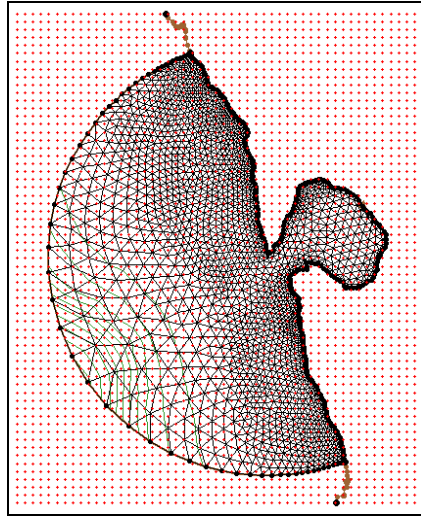


Figure 19-2 The completed finite element mesh.

At this point, the display is quite cluttered with all the data that has been created. Some of the visible objects can be hidden using the *Project Explorer*. To hide the scatter data and map data, uncheck the toggle boxes next to those objects in the *Project Explorer*.

To change the display settings use the *Display Options*  dialog by selecting the macro or right clicking on one of the model folders in the *Project Explorer* and choosing *Display Options*.

1. Switch to the *2D Mesh* tab and turn off everything except the *Elements*, *Contours*, and *Nodestrings*.
2. Click the *Contour Options* tab. Change the *Number of Contours* to 20. Change the *Contour Method* to *Color Fill*.
3. Click the *OK* button.

After the display is refreshed, you will see contours of water depth with the elements drawn on top of those. You can clearly see that as the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.

19.6 Model Control

When creating a *CGWAVE* model, the boundary conditions are wave amplitude, phase, and direction. To define these *incident wave conditions*:

1. Click on the *Mesh* object in the *Project Explorer* to make it active and select *CGWAVE | Model Control*.
2. Set the *Incident Wave Conditions*: *Direction* = 30.0, *Period* = 20.0, and *Amplitude* = 1.0.
3. Set the *Output Echo Frequency* to 1 and the *Maximum Iterations* to 500,000.
4. CGWAVE uses a 1-d file. The 1-d parameters must be set in this dialog and the 1-d depths extracted. By default, an ideal spacing is computed and the # of 1-d nodes is set to run to 1.5*radius away from the coastline. We can leave these defaults. Click on the *Extract 1-d Depths* button to extract the values.
5. Choose the elevation (Z) function to extract from and click *Select*.
6. Click the *OK* button to exit the *CGWAVE Model Control* dialog.

19.7 Renumbering

The mesh needs renumbering before being saved. To do this:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the blue ocean nodestring by clicking in the box on the nodestring.
3. Select *Nodestrings | Renumber*.

19.8 Saving the CGWAVE Data

CGWAVE uses a geometry file and the 1-d file mentioned above to run an analysis. This file consists of two lines that run perpendicular from the coastline to the extents of the domain. The 1-d file is generated automatically by SMS using the active scatter set. The file contains depth information on both sides of the domain. To save these files:

1. Select *File | Save New Project*, and enter the name *indianaout*.
2. Push the *Save* button.

19.9 Running CGWAVE

CGWAVE can be run from SMS. To run CGWAVE:

1. Select *CGWAVE | Run CGWAVE*.

2. SMS saves the location of the *CGWAVE* executable as a preference. If this preference is defined, the model will launch. If the preference is undefined, SMS shows a message that the *hivel2d* executable is not found, click the *File Browser*  button to find the *CGWAVE* executable and Click the *OK* button to run the model.

For this simulation, *CGWAVE* should finish in a couple of minutes. When the simulation is finished, the file *indianaout.cgo* will contain the *CGWAVE* solution data. You can open this file and view the solution using the *File|Open* command. One of the model parameters for *CGWAVE* is wave breaking. If this option is on, the model will compute how the wave break. If not, you can still approximate the breaking by selecting the option to break the waves as you read the solution file. As you open the file, SMS will translate the wave output into data sets that can be visualized. These include wave phase, height, direction and sea surface, pressure and particle velocity at three locations in the water column, and a time series of wave surface and wave velocity over a wave cycle.

NOTE: If *CGWAVE* does not run, you may have an older version of *CGWAVE*. Open the *indianaout.cgi* file in a text editor and change the first few lines from:

```
%number of characters in title &
%number of terms in the series &
%number of iterations for checking convergence &
%maximum iterations &
%maximum iterations for nonlinear mechanisms &
%maximum connectivity &
      12      35      1      500000      1000      8
```

to:

```
%number of characters in title &
%number of terms in the series &
%number of iterations for checking convergence &
%maximum iterations &
%maximum connectivity &
      12      35      1      500000      8
```

19.10 Conclusion

This concludes the *CGWAVE* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File | Exit*. If prompted to confirm, click the *Yes* button.

Modeling with BOUSS2D

20.1 Introduction

This lesson will teach you how to use the interface for *BOUSS-2D* and run the model for a sample application. As a phase-resolving nonlinear wave model, BOUSS-2D can be used in the modeling of various wave phenomena including shoaling, refraction, diffraction, full/partial reflection and transmission, bottom friction, nonlinear wave-wave interactions, wave breaking and dissipation, wave runup and overtopping of structures, wave-current interaction, and wave-induced currents. This example will step you through the process of setting up and running a simulation. Data from Barbers Point, HI will be used. All files needed in the tutorial are included in the *tutorial\tut20* directory

To begin with, open the background data for this project. This includes images, bathymetry data and coastline data for the south west corner of the island of Oahu. To open the files:

1. Select *File | Open* and select the file “topo1.jpg” from the *tutorial\tut20* directory. Since the directory also includes a “.wld” file, SMS will display the image. Should SMS ask if image pyramids are desired, select the toggle to not ask this question again and then click “yes”.
2. Repeat the process to open the file “topo2.jpg”.
3. Select *File | Open* and select the file “BP_bathy_filtered.pts”. This file includes depth values obtained from a local survey in the Barbers Point harbor and the immediate coastal region outside the harbor.

4. Select *File | Open* and select the file “BP_coast.cst”. This file contains the coastline definition for the entire island of Oahu.

20.2 Specifying Model Units

The images came from <http://teraserver.homeadvisor.msn.com> and are therefore registered to the UTM NAD 83 coordinate frame. The bathymetry has been transformed to be relative to this coordinate frame. To tell SMS what coordinate system this data is related to:

1. Select *Edit | Current Coordinates*.
2. Make sure the *Horizontal System* is set to *UTM NAD 83* set the zone to “Zone 4” and set the *Horizontal* and *Vertical Units* are set to *Meters*.
3. Click *OK* to exit the dialog.

The data after reading in the bathymetric data should appear as shown in Figure 20-1. After the coastline is read in, the data should appear as shown in

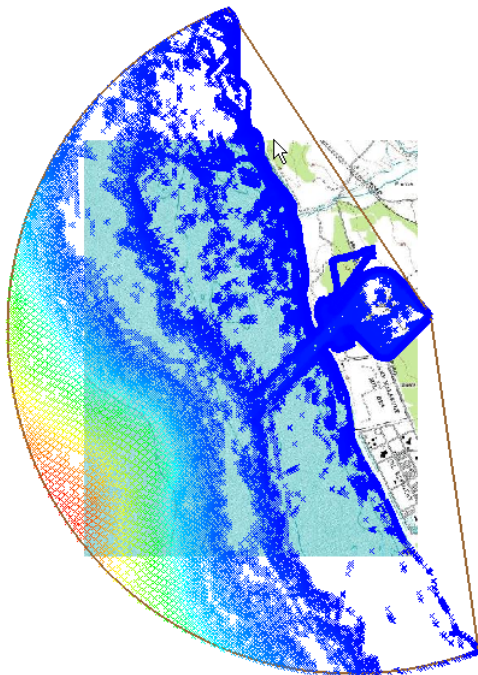


Figure 20-1. Bathymetry and images for project area.

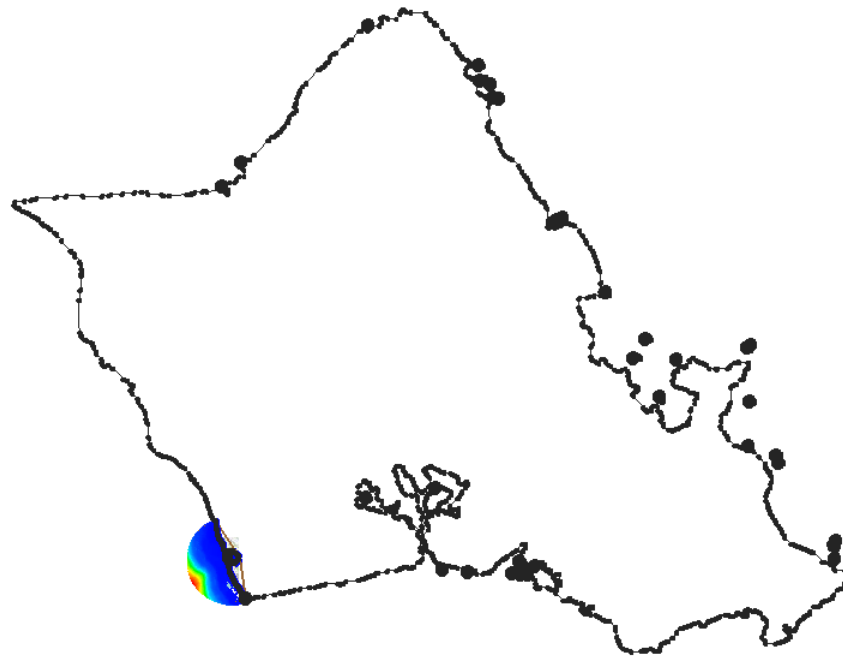


Figure 20-2. Coastline of Oahu with bathymetric data.

20.3 Trimming the Coastline

We have more coastline here than is needed, which includes many other harbors, land features, and islands unrelated to Barbers Point harbor. The user may use the following steps to trim the coastline to the area involved:

1. Zoom into the area being modeled.
2. In the Map Module , select the Create Arc tool .
3. Click on the coastline just north of the data (P1 in Figure 20-3), then click inland (P2) and then on the coastline east of the simulation area (P3).
4. Switch to the Select Arc Tool  and select the coastline away from area of interest and hit the Delete key to eliminate this arc (Arc to delete in Figure 20-3).
5. Frame the display and drag a box around the island arcs and delete them. (This is easily done by dragging a box around the islands on the north side of Oahu and deleting them, and then dragging a box around the islands on the east side.)

Build a polygon to represent the land around Barbers Point by selecting Feature Objects|Build Polygons.

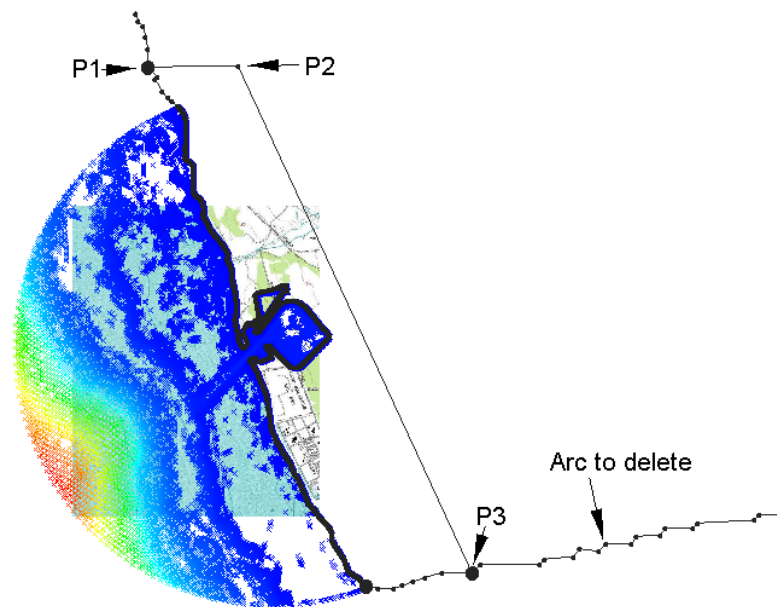


Figure 20-3. Arc bisecting area around simulation from island.

6. Select the Select Polygon tool , and select the land polygon.
7. Choose Feature Objects|Select/Delete Data ... This brings up the dialog shown in Figure 20-4. Set the options to delete triangles inside the polygon and click OK.

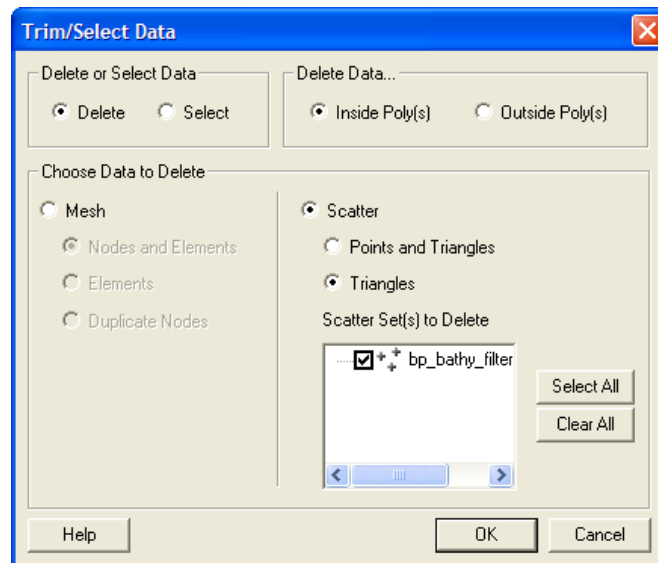


Figure 20-4. Trim/Select Data Options

We now have a surface that represents the seabed around the region of Barbers Point harbor in Hawaii. The next step is to create a computational grid for *BOUSS-2D*.

20.4 Creating the Grid

The computational domain of *BOUSS-2D* is a Cartesian grid that could be defined with three clicks. In this example, we shall provide instructions to ensure consistency. To create the grid, follow these steps:

1. Zoom into the harbor area as shown in Figure 20-5.

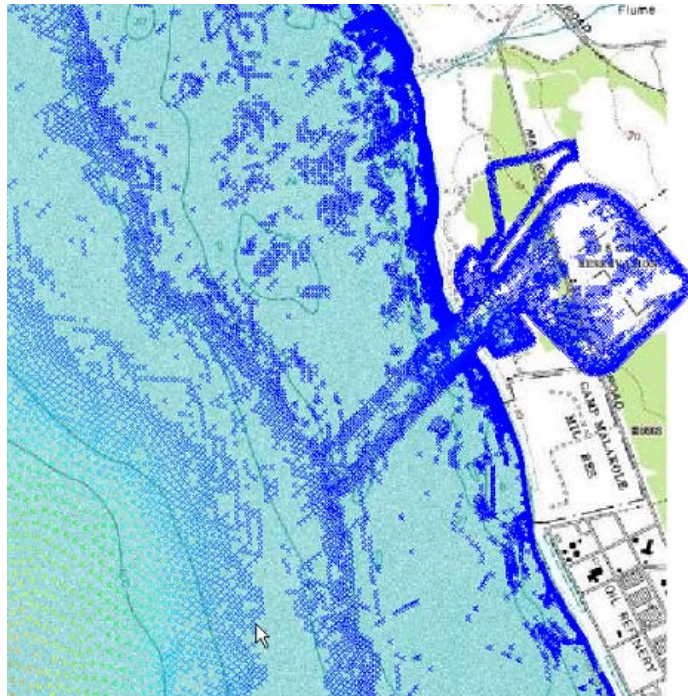


Figure 20-5. Zoomed view of harbor area.

2. Make sure the *2D Cartesian Grid Module*  is the current module and that *BOUSS-2D* is the active model. User should see a *BOUSS-2D* menu at the top menu bar in *SMS*. If this is not the case, select Data|Switch Current Model and select *BOUSS-2D*.
3. Select the Create Grid tool . Click near the point P1 (Figure 20-6). An exact location can be specified later. The coordinates of the cursor are displayed on the lower left corner of the window.
4. Move the mouse toward shore and inland. At bottom of the window, the size of resulting grid is displayed. Click on point P2 (Figure 20-6), approximately 2500 meters from the first point.

Move the mouse up to include harbor approximately 2000 meters from second point and click in the area of point P3 (Figure 20-6).

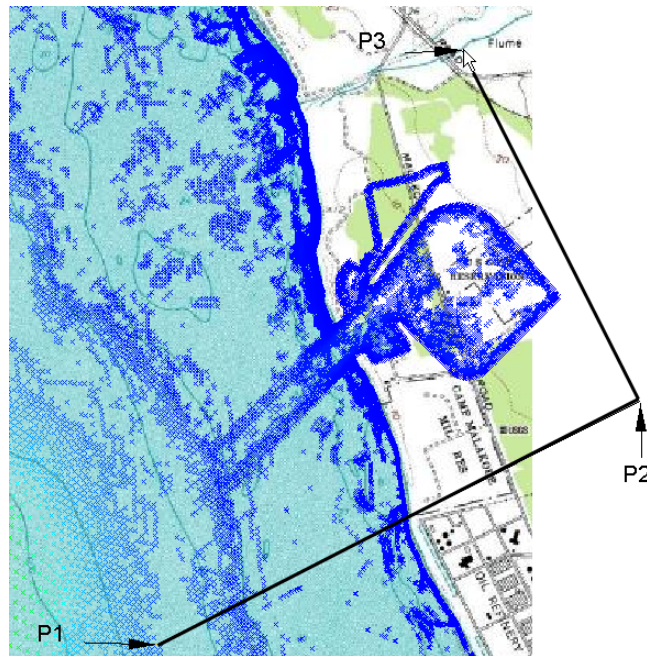


Figure 20-6. Click points for defining the grid.

Map -> 2D Grid

Grid Geometry

Origin:

X: 590100.00000 U: 2500.00000 m

Y: 2356750.0000 V: 2000.00000 m

Angle: 23.00000 deg

Cell Options

☒ Use Grid Frame Size

Cell Size: Delta U 10.00000 m Delta V 10.00000 m

Columns 250 # Rows 200

Depth Options

☐ Constant 0.000 m

☒ Interpolated elevation [Z]

☐ Current

☐ Constant

Current x: 0.000 m/s

Current y: 0.000 m/s

☒ Interpolated

Help OK Cancel

Grid Parameter	Value
Grid Origin X	590100.0
Grid Origin Y	2356750.0
Angle	23.0°
Grid Size U	2500 m
Grid Size V	2000 m
Cell Size Delta U	10 m
Cell Size Delta V	10 m

Figure 20-7. Parameters to create the grid.

- After the third click, SMS will display a dialog to create the grid (Figure 20-7). For consistency in this exercise, modify the grid parameters in this

dialog to match the values in the dialog as shown. The cell sizes are dependent on the wavelength. The Courant number that controls the stability of numerical solution in *BOUSS-2D* is used to determine time step size from the cell size. The cell size is often expressed as a fraction of the wavelength. Care should be exercised in selecting cell sizes. For tutorial purposes, 10 m cells are used in this example. In the *Depth Options*, click the button labeled “elevation (Z)”. This will bring up the Interpolation dialog. Set the extrapolation elevation on the left side to be 1 meter. This assures that the land will be treated as land. This step would not be required if survey data included points on the shore with positive elevations. Click the OK buttons on both the dialogs to create the grid.



Figure 20-8. Resulting grid.

Figure 20-8 shows the resulting grid boundary. The computational (ocean) area is outlined in the darker colors. The area on the right (land) is inactive, and model calculations are not performed in the land regions.

20.5 Defining the Wave Maker

The *BOUSS-2D* wave maker must be positioned along a straight line, called here as “cell string” in SMS, at a desired location where depth is ideally constant. The above grid creation process generates cell strings along the edges of the computational area. Cell strings can also be created manually to specify the location of structures, wave-makers, and areas where damping and/or porosity layers may be necessary. To define a wave maker, follow these steps:

1. Select the Select Cell String Tool .
2. Select the cell string (in red) on the left side of the grid as shown in Figure 20-9.
3. Select BOUSS-2D|Assign BC command. Select the *Wave Maker* radio button and click on the options button. This brings up the *BOUSS-2D Wave Generator Properties* dialog. On the left side of this dialog select the type of wave to simulate, i.e. irregular unidirectional waves. Select the defaults to synthesize the time series and generate them from parameters. Specify the series duration as 750.0 seconds in this example. On the right side, select the spectra type to be JONSWAP with the option to specify significant wave height and peak period values, H_s and T_p . Enter values of significant wave height and peak period as 3.0 m and 15.0 sec. respectively. Click OK to close both the dialogs.
4. As SMS generates the wave maker, it may find that the offshore edge of the grid is not of constant depth, warning the user about this situation. At this point the user is given the option of extending the grid to a constant depth to improve model stability. Click the “Yes” button to extend the grid to complete properly defining the wave maker location.

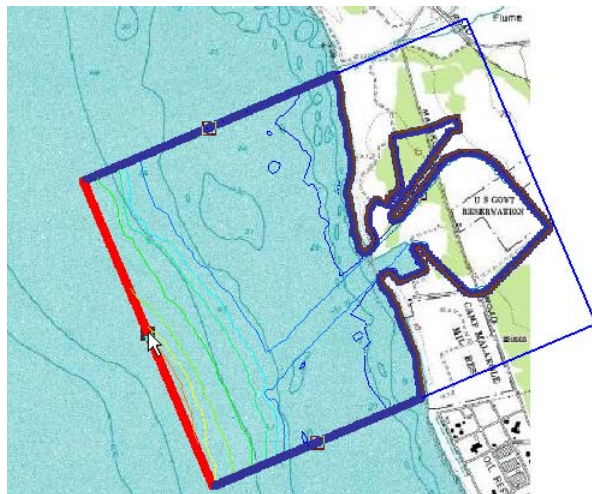


Figure 20-9. Wave maker cell string selected.

20.6 Defining Damping Layers

Damping layers may be specified along boundaries of the computation zone where waves need be absorbed. Damping layer prevent waves from reflecting back into the computational area that will affect incident waves. In this example, damping layers will be assigned to both lateral boundaries of the modeling domain. This simulates allowing waves to leave the domain. To do this:

1. Select the cell strings along the open sides of the grid between deep water and the coastline. To select two cell strings, click on the first and hold the SHIFT key to click on the second cell string.
2. Select BOUSS-2D|Assign BC... and select *Damping*.
3. Enter a width of 40 and a value of 1.0 and select OK. The width value for the damping depends on several factors including wave type and grid resolution.

In applications where the depth transition from water to land is not gradual, excessive reflection occurs and the model over predicts reflection from the coast. To prevent this, we place a damping layer along the coastline. The coastline damping layer may be defined as:

1. Select the cell string along the coastline.
2. Select BOUSS-2D|Assign BC... and select *Damping*.
3. Enter values for width and damping coefficients of 20 m and 0.5, respectively and select OK.

20.7 Other Model Parameters

The next step in preparing a simulation is to specify other model input parameters. Select BOUSS-2D|Model Control to specify Project Title as “Barbers Point Sample Run”. In the Input Data Sets, select “Use Cellstrings” for damping. For Porosity and Current, choose “None”. For Time Control, enter 1500 sec for duration of the run, and 0.2 seconds for the time step. For Animation Output, check out *Output WSE*, *Output Velocity* and *Override Defaults* toggles. Enter 1425.0 for the beginning time, 1500.0 for the ending time and 1.0 sec as the output interval to save an animation output for approximately the last 5 waves. Select the OK button to close the dialog box.

20.8 Saving and Running the Simulation

The final step before running a simulation is to save the files for *BOUSS-2D*. Select File|Save New Project and enter “BarbersPoint_Tutorial” for project name, and click OK. The simulation may now be launched by selecting BOUSS-2D|Run Bouss. A message will be displayed for default location of the model executable. Click on the folder icon to select the location of “Bouss2d.exe” file, and click on the *Run* button. SMS now launches *BOUSS-2D*. This simulation takes approximately 1 hr to run on a Pentium 4 processor (3 GHz clock speed) with a 0.2 sec time step. The model run time would increase to nearly 3 hrs for a grid using 5 x 5 m cells, and the run time would reduce to below 20 minutes for a grid with 20 x 20 m cells.

After the model run is complete, make sure the toggle to read the solutions is checked and click the *Exit* button.

20.9 Visualize Simulation Results

The model will create five solution data sets that include spatially varying results at the grid nodes. Three of these are steady state data sets that include the results for mean wave height, mean current direction and mean water level. The other two are temporally varying results, output at user specified intervals; they include computed water levels and currents (u and v). *BOUSS-2D* can save these results in two ways; five separate files, or a single binary file (HDF5 format). In this case, *BOUSS-2D* should have create the file “BarbersPt.h5” and this file should have been read in at the completion of the run. SMS creates a folder in the *Project explorer* containing all of the data sets.

To display a functional surface of the water surface:

1. Select the *Display|Display Options* command and turn off the cells toggles (both water and land) and turn on the *Functional Surface* toggle.
2. Click on the *Options* button right next to the toggle. This brings up the *Function Surface Options* dialog.
3. In the upper left corner select the *User defined data set* option, click the *Select* button and choose *WSE Animation* from the list of datasets.
4. Click the *Select* button in the *Select Data Set* dialog.
5. Use the *Choose Color* button to select a color for the functional surface, then click *OK* to close the *Functional Surface Options* dialog.
6. Back in the *Display Options* dialog, choose the *General* tab and set the *Z magnification* to 20.0.
7. Select the *Contours Options* tab and change the *Contour Method* to *Color Fill*, and click *OK*.

To give the surface some feature:

1. Select *Display|Lighting Options* and select *Use light source*.
2. Turn on the *Smooth features*, click on the upper right side of the sphere, and then click *OK*.
3. Select the *Depth* data set to contour on the bottom of the ocean.

4. Select Display/View/View Angle and enter a bearing of 40 and a dip of 25.

Figure 20-10 shows this functional surface of the water surface over the bathymetric surface.

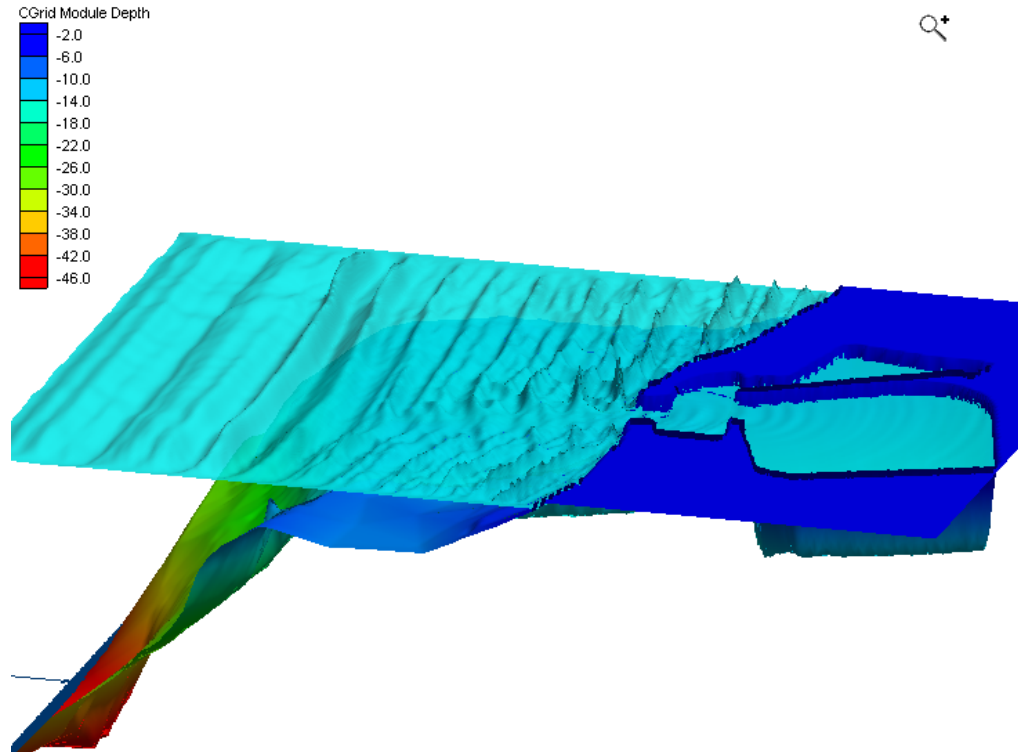


Figure 20-10. Functional surface of water levels over the bathymetry (magnified 20x).

All of the standard visualization methods described in the first few tutorials operate for the solutions generated by BOUSS-2D. Experiment to view the solutions.

20.10 Conclusion

This concludes the Modeling with BOUSS2D tutorial. You may continue to experiment with the SMS interface or you may quit the program. To quit SMS at this point:

- Select *File* | *Exit*. If prompted to confirm, click the *Yes* button.

HEC-RAS Analysis

23.1 Introduction

HECRAS was developed by the U.S. Army Corps of Engineers Hydrologic Engineering Center. *HECRAS* performs a step backwater curve analysis for either steady state or transient conditions to determine water surface elevations and velocities.

23.2 Preparing the Conceptual model

The first step to creating a *HECRAS* Model is to create a conceptual model which defines the river reaches (layout and attributes), the position of cross-sections on those reaches (orientation and station values), bank locations, and material zones. The conceptual model will be used to create a network schematic inside the *1D River*

Hydraulic Module .

We will create the conceptual model from an USGS quad map and scattered bathymetric data. To load this information, open the files *LeithCreek.img* and *elev.sup*.

The scatter points clutter the screen, but we want to know where they are so we don't create our conceptual model outside of domain of our bathymetric data. To better see the image and still know where the extents of our points are we will turn off our scatter points and turn on the scatter boundary. To do this:

1. Select the Display Options  button.
2. Select the Scatter tab.
3. Uncheck the Points toggle box.
4. Check the Boundary toggle box.
5. Click OK.

23.2.1 Creating the Coverages

We need to create a centerline coverage for our reaches and a cross-section coverage for our cross-sections. These will form the core of our conceptual model.

1. Set the current module to be the *map*  module.
2. Select *Feature Objects | Coverages...*
3. Change the name of the default coverage to “centerline” and set its type to be *1D Hyd Centerline*.
4. Create one new coverage by clicking the *New Coverage* button .
5. Change the name of the new coverage to “Cross-section” and change its type to *1D Hyd Cross-section*.
6. Make sure that the centerline coverage is active by clicking the active box in the centerline coverage row.
7. Click the *OK* button.

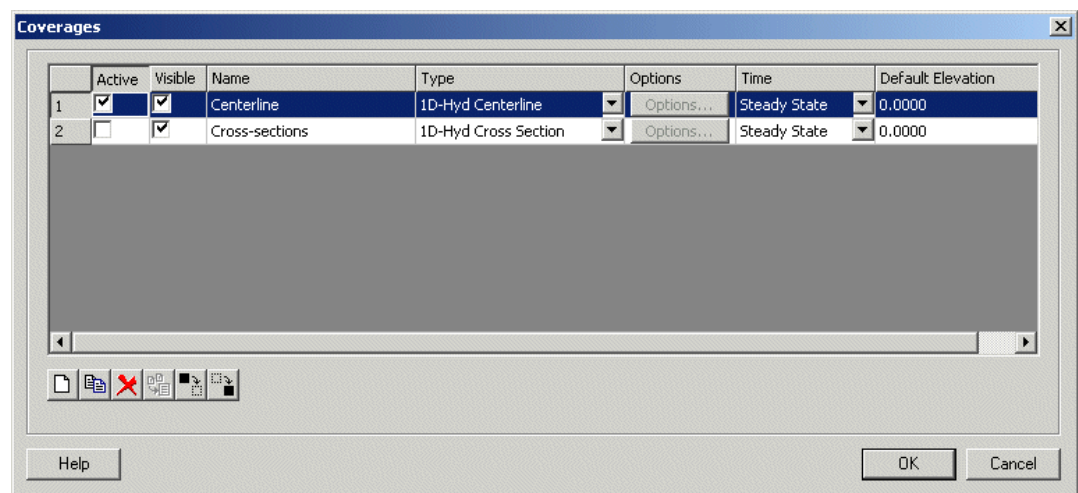


Figure 23-1 The coverage dialog.

23.2.2 Creating Centerline and Bank arcs

Before creating feature arcs we will tell SMS that we will be using the HECRAS Model. Select *Feature Objects | Set 1D Model* and choose *HECRAS*.

Centerline arcs are used to define the locations and lengths of the study reaches and assign their attributes. We will have a centerline following the main channel of Leith Creek as well as the tributary on the West. As the flows below the reservoir in the tributary on the East of the Leith Creek are small, we will disregard that reach in our simulation. To create the centerline arcs:

8. Select the *Create Feature Arc* tool .
9. Following the pattern in Figure 23-2, create the centerline of the main channel from upstream to downstream by clicking points on the centerline one at a time. Double-click the last point to indicate that it is the end of the centerline.
10. Create the arc for the west tributary upstream to downstream by clicking points on the centerline. Create the last point where the tributary meets the main channel by clicking on the main channel centerline. This splits the centerline of Leith Creek into two reaches.

This defines the centerline of the three reaches, on two rivers, that we will model in this simulation.

Bank arcs are used to define the locations of the banks and the overbank distances. The next step is to create bank arcs along both sides of each centerline arc. Using the

Create Feature Arc tool  create new arcs where you estimate the bank locations to be (based upon contours on the background image). New arcs are assigned by default to be centerline arcs. Change them to bank arcs by following these steps:

Select the bank arcs.

11. Choose *Feature Objects | Change Arc Type*.
12. Change the type combo box to *bank arcs*.
13. Click *OK*.

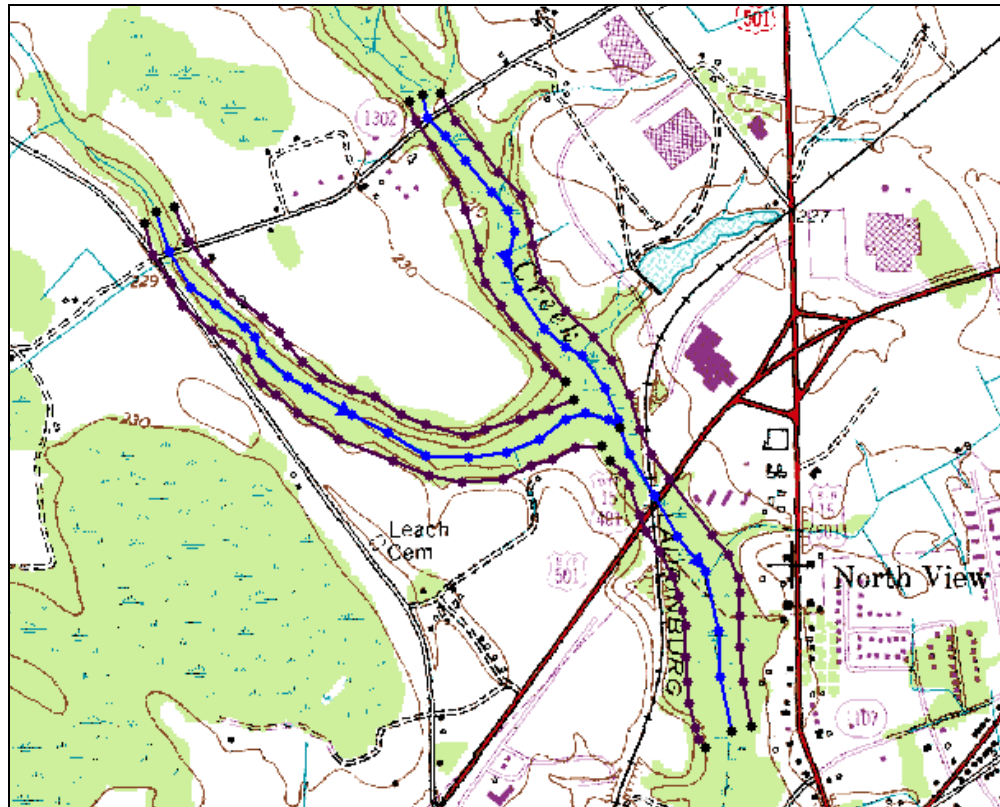


Figure 23-2 Centerline and bank arc placement

The background image is no longer necessary. To turn it off:

1. Select the *Display Options*  button.
2. In the *Image* section, uncheck the *Display Image* box by clicking on it.
3. Press OK.

23.2.3 Naming the Centerline Arcs

Reaches are stream sections where the flowrates and other hydraulic conditions are assumed to be constant. A river can include one or more reaches, but a river is not allowed to fork in HECRAS. HECRAS has the ability to model multiple rivers (flowpaths). To assign names to our rivers and reaches:

14. Using the *Select Feature Arc* tool,  double-click the uppermost reach in the main channel.
15. Set the *River Name* to “Leith Creek”.
16. Set the *Reach Name* to “Upper Main”.

17. Click OK.
18. Repeat steps 19 to 22 for each reach in the map as shown below. (Note: For the Lower Main reach you can choose “Leith Creek” from the river name combo instead of typing it in.)

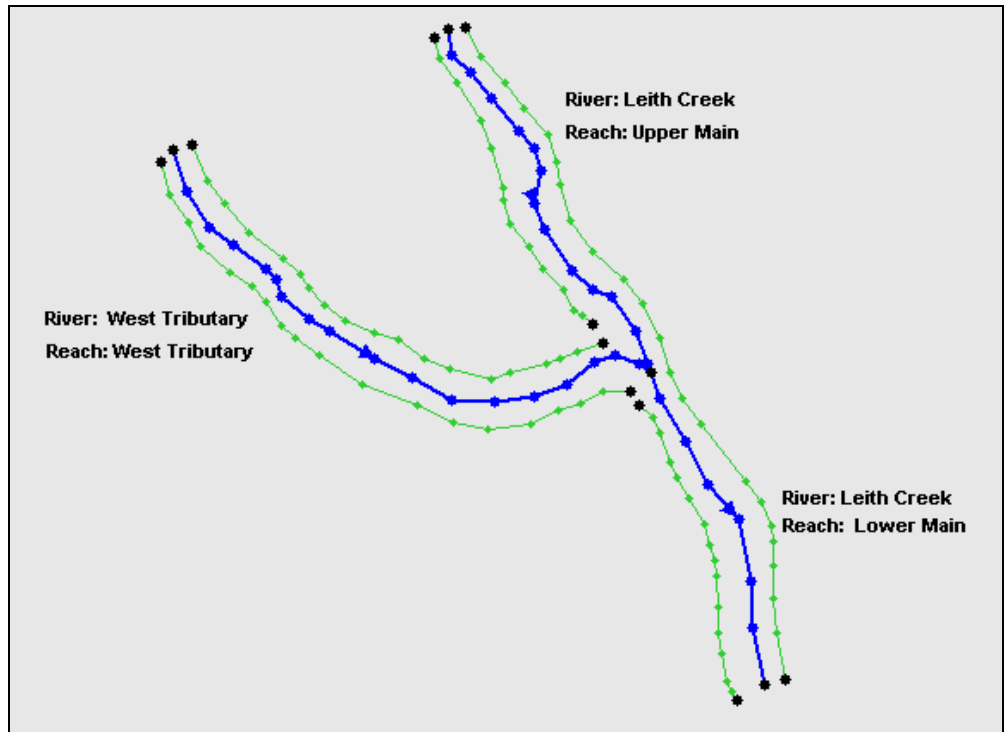


Figure 23-3 River and Reach names

23.2.4 Creating Land Use Coverage

One of the properties HECRAS uses is roughness values. We will designate materials to different areas of our model. Later we will assign each material a roughness value. The material zones are stored in SMS in a coverage of type Area Property. To load the materials data:

19. Open the file *Materials.map*.
20. Make sure the newly created Area Property coverage “materials” is active.
21. Click on the *Display Options*  button.
22. Turn on the *Polygon Fill* and *Legend* options.

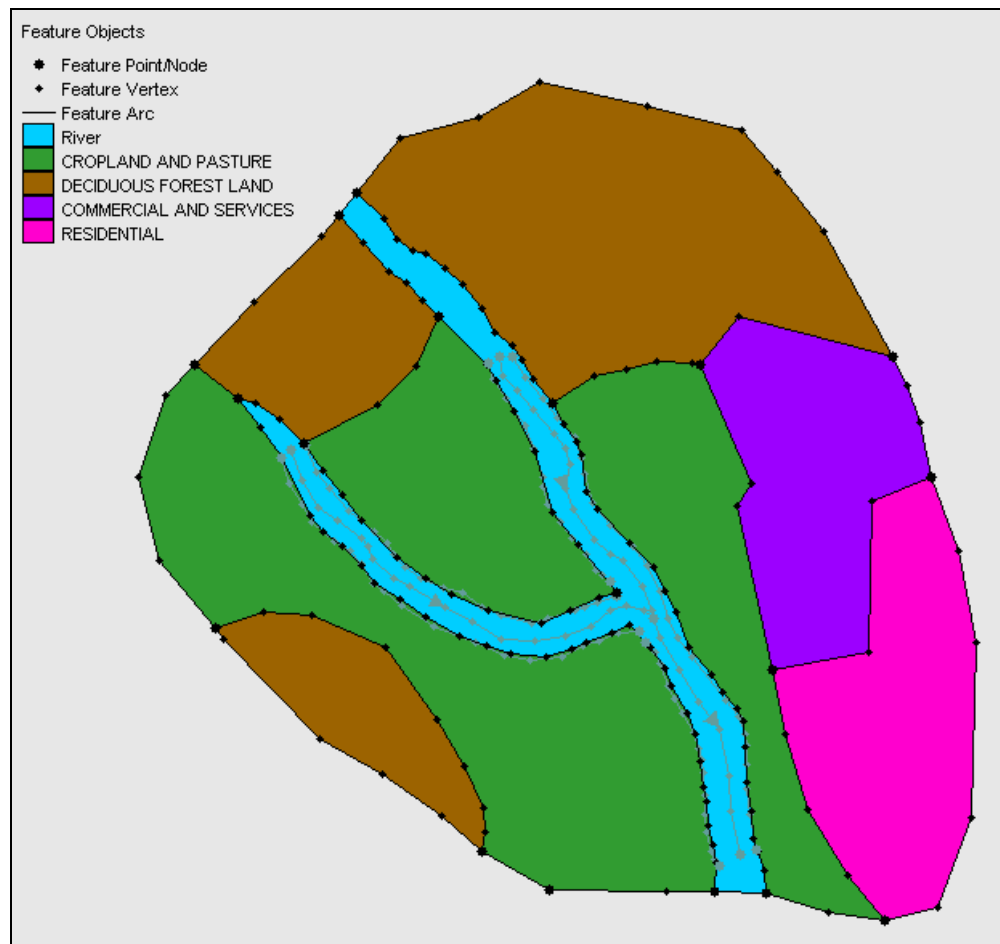


Figure 23-4 Materials used in HECRAS Simulation.

23.2.5 Creating the Cross-Sections

HECRAS associates most of its model data with cross-sections and generates solutions or output at the cross-sections. Therefore, cross-sections are the most important part of the map. *HECRAS* requires at least two cross-sections on each reach.

23. Set the current coverage in the *Coverage* combo-box on the top of the edit window to “Cross-Section.”

24. Select the Create Feature Arc tool .

25. Create at least two cross-sections on each reach by clicking a point on one side of the reach then double-clicking a point on the other side of the reach as shown in Figure 23-5.

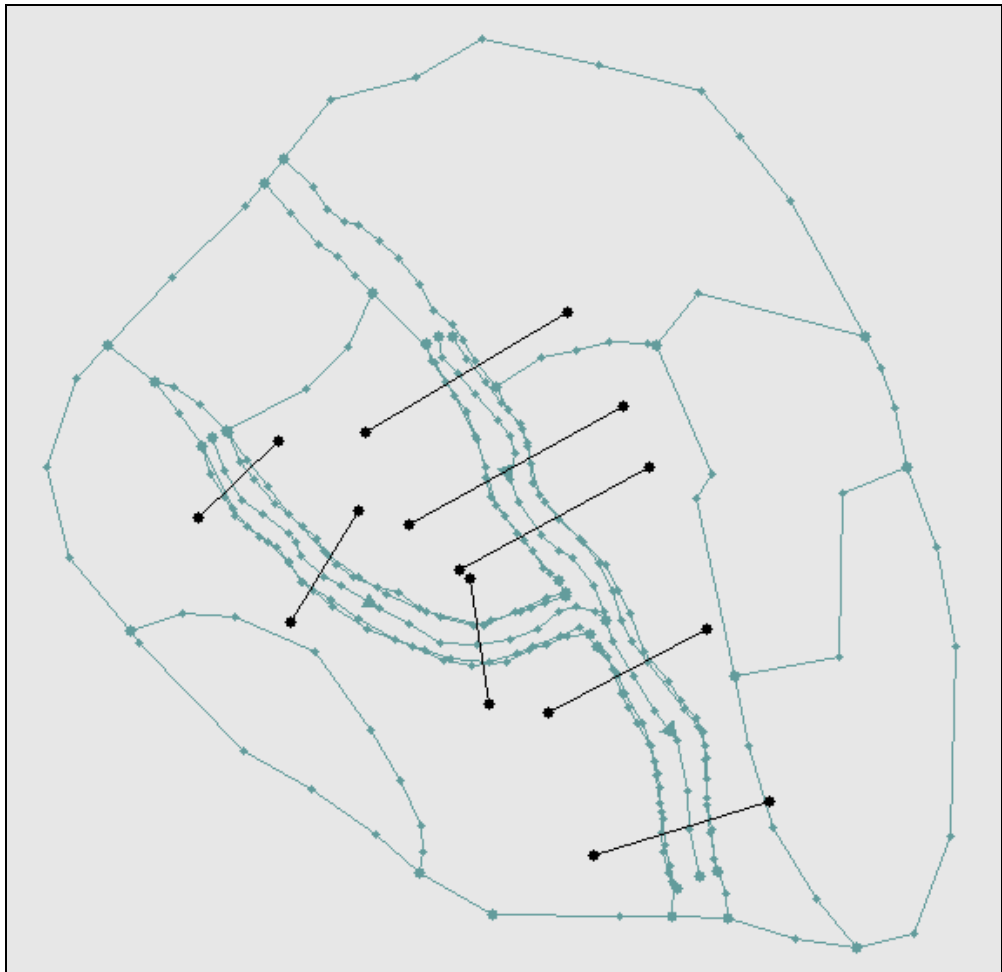


Figure 23-5 Cross-section coverage

23.2.6 Extracting Cross-Sections

In the cross-section coverage, all arcs are cross-section arcs. Their position and orientation define the location of the cross-sections in the system, but as of yet, they do not have any data assigned. We want to assign elevation data, materials, and point property locations to the cross-sections. This information will be extracted from the scattered data set (and its TIN), the area property coverage, and the centerline coverage. To extract this data:

1. Select *Feature Objects | Extract Cross Section*.
2. SMS will set the defaults to use the centerline coverage to generate point properties and to use the area property coverage “materials” to define material zones. Click *OK*.
3. SMS will prompt for a location to save the cross-section database. Enter the file name *xsecs*.

Each arc now stores a link to a cross-section database record which contains xyz data, materials properties, bank locations, and thalweg locations. To view the information at a cross-section:

26. Click on the *Select Arc Tool* .
27. Double click on any cross-section. This brings up the *River Cross Section Attributes* dialog.
28. Make sure that the reach name is assigned correctly.
29. Click on the *Assign Xsec* button. This brings up the *Assign Cross Section dialog*, which is used to view the current cross section shape and select a different cross-section from a cross-section database if desired.
30. Click on the *Edit* button. This brings up the *Cross-Section Attributes* dialog. This dialog can be used to view and/or edit the cross-section.
31. Click on the *Line Props tab* to view the materials that are assigned to the cross-section.
32. Click on the *Point Props tab* to view the locations of the left bank, right bank, and thalweg.
33. Click *Cancel* until all the dialogs are closed.

23.3 Creating the Network Schematic

SMS interacts with HECRAS using a HEC-GeoRAS geometry file. This file contains the cross-sectional data used by HECRAS in addition to three dimensional georeferencing data. To create this geometry file the conceptual model must be converted to a network schematic diagram in the *1D Hydraulic Module*. To convert the conceptual model to a network schematic:

34. Make sure you are still in the Map Module .
35. Set the current coverage in the coverage combo-box on the top of the edit window to “*Centerline*.”
36. Select *Feature Objects | Map -> Schematic*.
37. Switch to the *1D River Hydraulic Module* .

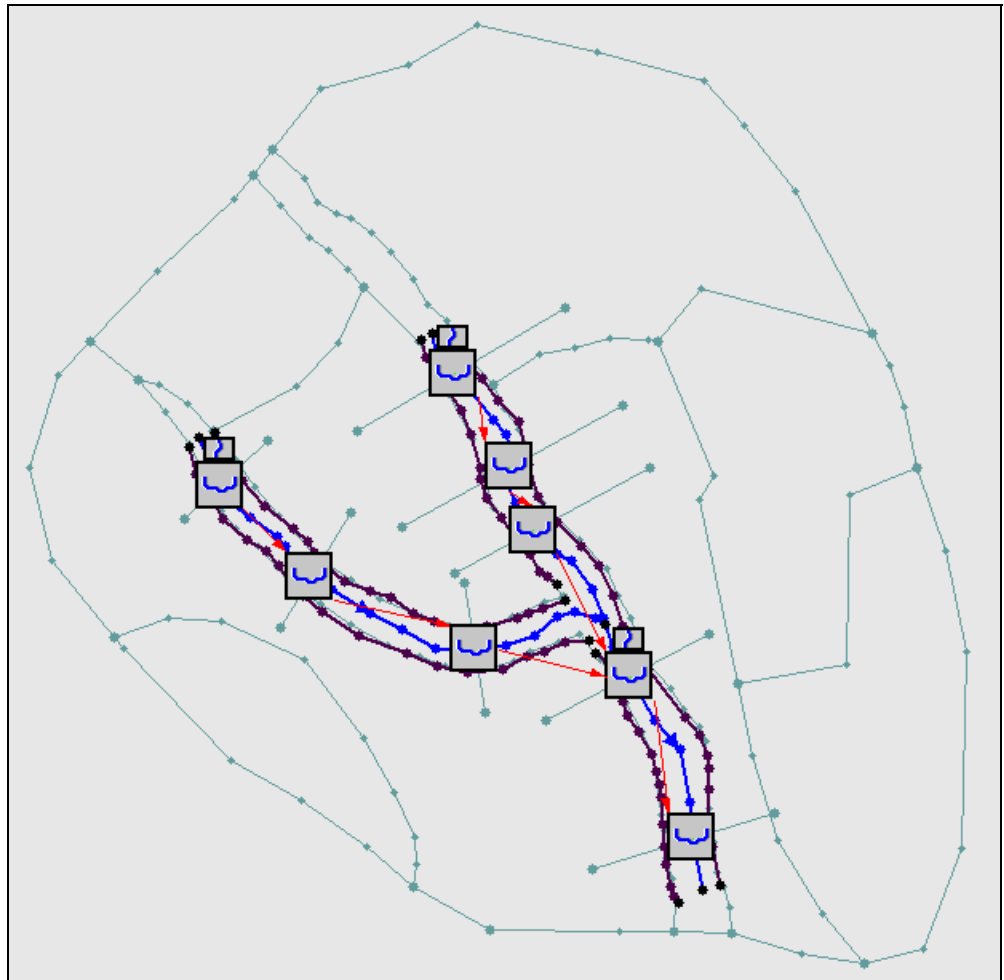


Figure 23-6 Schematic diagram

Now SMS includes two separate representations of the data. The first you created as a conceptual model which is stored as a series of coverages. The second is a numeric model stored as a schematic of cross sections organized into reaches. Modifications to the network schematic that can be used by HECRAS can be made directly in the *1D River Hydraulics Module*, or indirectly by editing the conceptual model in the *Map Module* and mapping to a new network schematic.

HECRAS needs Manning's roughness values for the materials found in the cross-section database. The roughness values are stored as part of the 1D model in the *1D River Hydraulics Module*. To specify the roughness values for the each of the materials:

38. Select *HECRAS | Material Properties*.
39. Enter the roughness values for each material as shown in Figure 23-7.

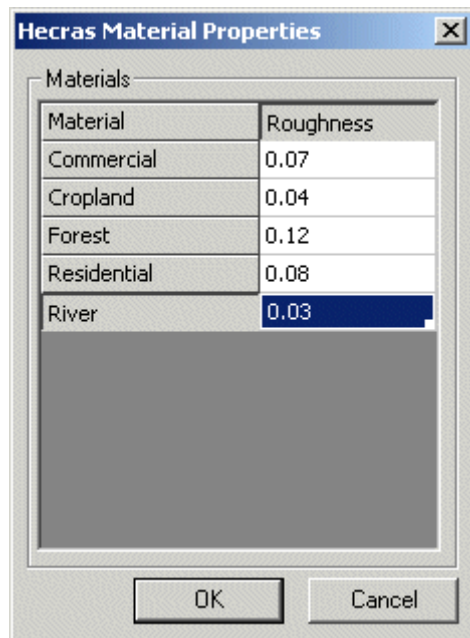


Figure 23-7 HECRAS Material Properties Dialog

40. Click OK.

Now we need to tell HECRAS which set of line properties in the database should be used as material types. To do this:

41. Select *HECRAS | Model Control*.

42. Select the line property name that stores the roughness values for the cross-section database. In this case, the line property is named Materials which came from our Area Property coverage “Materials.”

43. Click OK.

23.3.1 Creating the Geometry import file

Now that the simulation has been setup we need to create the geometry file. To create this file:

44. Select *File | Save As...*

45. Change Save type to *HECRAS Import File*.

46. Name the file *LeithCreekHecras.geo*.

47. Click *Save*.

23.4 Using HECRAS

In HECRAS we need to read in our geometry file, assign boundary and flow conditions, setup and run the simulation, and export the results for post-processing in SMS.

The geometry that we exported to use in HECRAS is imported through the geometry editor inside of HECRAS. To import this file:

1. Startup HECRAS.
2. Create a new project using *File | New Project*.
3. Give the project a title and a filename and click *OK*.
4. Choose *Edit | Geometric Data* to bring up the geometry editor.
5. Select *File | Import Geometric Data | GIS Format*.
6. Browse to the file *LeithCreekHecras.geo*.

HECRAS has a tool that will filter points that are too close together to run an analysis. To filter the points (still inside the Geometric Data editor):

1. Select *Tools | Cross Section Points Filter*.
2. Click on the *Multiple Locations* tab.
3. Choose *Leith Creek* from river combo-box.
4. The River Sta. box ought to have *All Reaches* highlighted. Click on the arrow to the right of the box to select all cross-sections.
5. Click *Filter Points on Selected XS*.
6. Click *Close*, and then *OK*.
7. Close the Geometric Data Editor by choosing *File | Exit Geometry Data Editor*.

The next step is to define the flow and boundary conditions for our reaches. To define this information:

1. Select *Edit | Steady Flow Data* from the menu.
2. For Profile 1 (PF 1) enter 4000 for the Upper Reach, enter 1000 for the West Tributary, and 5000 for the Lower Reach.
3. Click on the *Reach Boundary Conditions* button.

4. For our analysis we are going to have HECRAS compute normal depths at boundaries of our model. To do this, for each of the blank boxes in the spreadsheet, select the box and click on the *Normal Depth* button. Enter the following values for the slopes of each reach: 0.003 for the upper reach, 0.001 for the lower reach, and 0.005 for the tributary.
5. Click *OK*.
6. Click *Apply Data*.
7. Select *File | Exit Steady Flow Editor*.

We are now ready to run the steady flow analysis. We first need to set an option to set flow distribution locations so that velocity profiles will be computed. To set this option and perform the analysis:

1. Select *Run | Steady Flow Analysis* from the menu.
2. Click on *Options | Flow Distribution Locations*.
3. Change the Global subsections to 3 in each of the three fields (LOB, Channel, and ROB).
4. Click *OK*.
5. Click *Compute*. This runs the 1D analysis.
6. Close the Steady Flow Analysis Dialog.

Now that the simulation has been completed, we need to export the data for post-processing inside SMS. To export the data:

1. Choose *File | Export GIS data*.
2. Click *Browse* and choose to save the file as “*LeithCreekOut*” in your project directory.
3. Select *Export Velocity Distribution Information where available* and *Export User Defined Cross Sections*.
4. Click *Export Data*.

Exit HECRAS and go back to SMS.

23.5 Post Processing

SMS can create cross-section and profile plots of the study reach. This section will demonstrate these capabilities. With the project file still loaded, open the file

LeithCreekOut.RASexport.sdf. This will load the datasets from HECRAS into memory.

23.5.1 Profile Plots

First we will create a profile plot to see the big picture of what is happening. To create a profile plot that shows the elevation and water surface elevation of all the cross-sections along the main channel:

1. Click on the *Plot Wizard* tool .
2. Select the plot type *1D Hydraulic Profile* and click *Next*.
3. Change selected reaches to specified reaches and check the Upper and Lower Reaches.
4. Turn on the display of minimum values and turn off the display of average values.
5. Under Datasets choose specified data sets and turn on *Elevation* and *Water Surface Elevation*.
6. Click *Finish*.

The plot shows the station along the x-axis and the elevation and water surface elevation along the y-axis.

23.5.2 Cross Section Plots

We will now create a cross-section plot to look at velocity distributions at various cross-sections. To create this plot:

1. Click on the *Plot Wizard* tool .
2. Select the plot type *1D Hydraulic Cross Section* and click *Next*.
3. Leave the Cross-sections as selected cross-sections and the datasets as the active data set. Click *Finish*.
4. Using the scalar combo box in the edit window, change the dataset to *velocity*.
5. Using the *select Cross-section tool*  select a cross-section in the schematic.

Look at the velocities of various cross-sections by selecting different cross-sections.

23.5.3 Post Processing Experimentation

You can now experiment with the plot tools to visualize the other data computed by HECRAS.

23.6 Conclusion

This concludes the HEC-RAS Analysis tutorial. If you wish to exit *SMS* at this point:

- Choose *File* | *Exit*.

This lesson runs through the process of creating a GENESIS simulation with an STWAVE grid to track a changing coastline.

24.1 Setting Current Coordinates

Before running any simulation, all data must be in the same physical coordinate system. The data for this tutorial are in State Plane coordinates, NAD 1983 survey, Massachusetts Mainland zone. The units are meters. To set the coordinates in SMS:

1. Select *Edit | Current Coordinates*.
2. Set the *Horizontal System* to *State Plane NAD 83 (US)* and set the *St. Plane Zone* to *Massachusetts Mainland - 2001*.
3. Change the *Units* for the *Horizontal System* and *Vertical System* to *Meters*.
4. Click *OK*.

24.2 Opening Bathymetric Data

The bathymetric data points are listed in the file STW_bath.xyz. These were taken from a survey and were converted to State Plane zone 2001 outside SMS. To open these points:

1. Choose *File | Open* and open the file “STW_bath.xyz”.
2. Make sure the file is detected as *Delimited* with *Space* and *Tab* options and click the *Next* button.
3. Verify the *SMS Data Type* is *Scatter Set* and click the *Finish* button.

The data are shown in Figure 24-1. The coastline is on the left side so that the point spacing increases in deeper ocean.

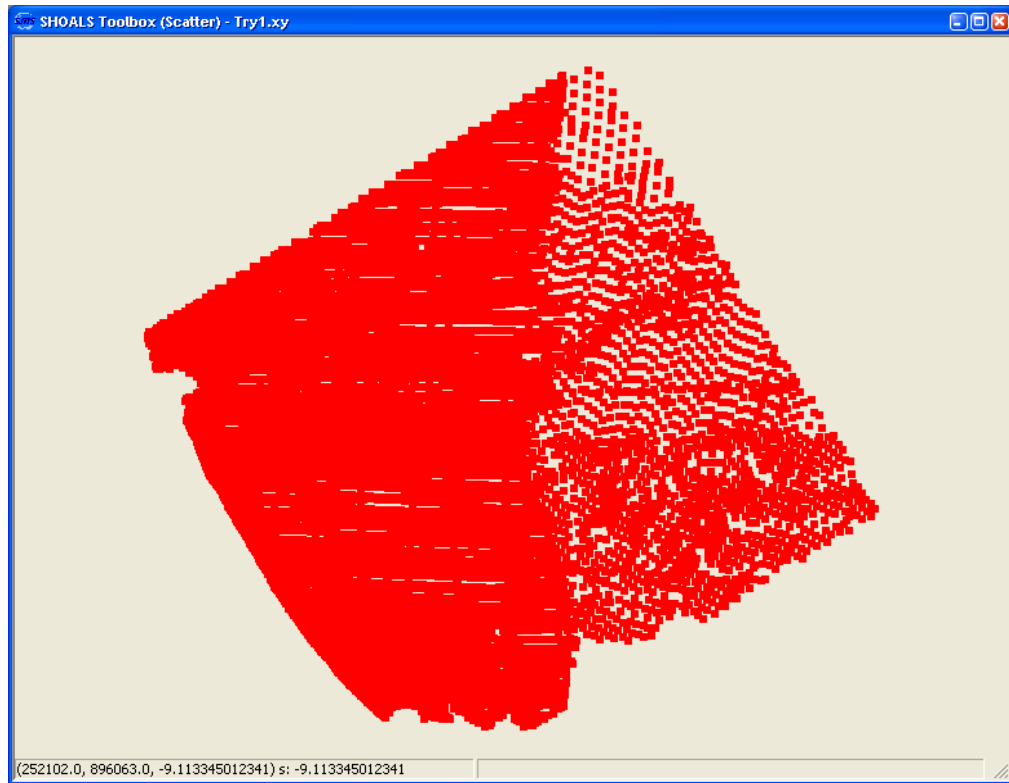


Figure 24-1. Bathymetric data for this lesson.

24.3 Converting Bathymetry to Depths

The data from the .xyz file are bathymetry (negative values in the ocean) but the STWAVE model deals with depths (positive values in the ocean). The bathymetry values need to be converted to depth values. To do this:

1. Make sure the *Scatter*  module is active.
2. Select *Data | Data Calculator*.
3. Click the minus (-) button and then double-click the “elevation” *Data Set* so that the *Expression* is “-c”.
4. Enter the name “depth” into the *Result* field.
5. Click the *Compute* button. The computed depth will appear in the *Data Sets* list.

6. Click the *Done* button to finish this operation.

The scatter points will now hold two sets of data: bathymetry and depth. The depth values will be used later when the grid is created.

24.4 Creating the STWAVE grid

Before using GENESIS, an STWAVE grid needs to be created over the ocean approaching the shoreline. To set to the active module as STWAVE:

1. Switch to the *Cartesian Grid*  module.
2. Select *Data | Switch Current Model*, set the active model to STWAVE and click *OK*.

The STWAVE grid is oriented with the origin on the ocean side and the positive X (Cell I) direction toward shore. For this model, the shore runs northwest-southeast with ocean on the east side. To define the grid:

1. Choose the *Create Grid Frame*  tool from the toolbox.
2. Create a grid frame by clicking first where the grid origin should be, then at the start of the shoreline, and finally at the shoreline end, as in Figure 24-2.

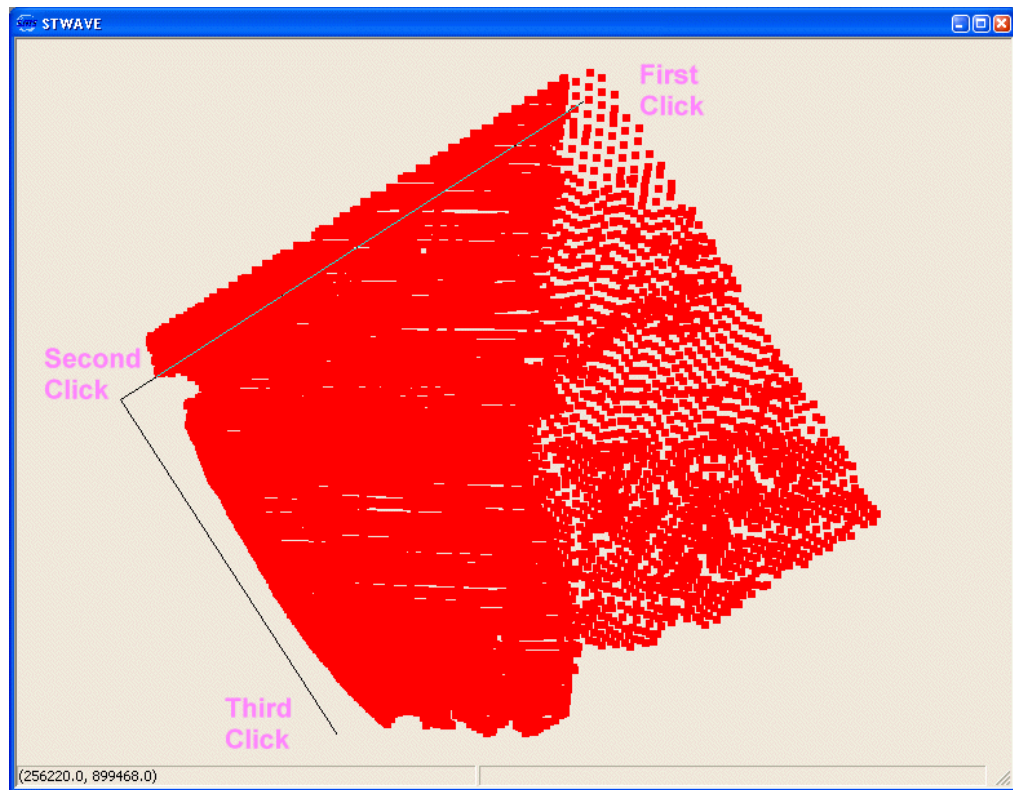


Figure 24-2. Creating the STWAVE Grid Frame.

When the grid frame is defined, the Map->2D Grid dialog will open, allowing the parameters to be modified. Many times exact parameters will not be known. But for this model, specific values will be entered to make sure the model is exactly the same as what the SMS developers used. To set these grid parameters:

1. Set the *X origin* to 256257.7955 and *Y origin* to 899379.9333.
2. Change the *Delta I* to 7376.16 meters and *Delta J* to 6339.84 meters.
3. Set the *Angle* to 212.0 degrees.
4. Use a *Cell Size* of 60.96 meters.
5. Make sure the *Interpolated* option is selected under *Depth Options* and that the button is named “depth”. If not, select this option, click the button and select the depth data set.
6. Click the *OK* button to create the grid.

SMS will create an STWAVE grid and overlay the scatter points, partially hiding the grid, as shown in Figure 24-3.

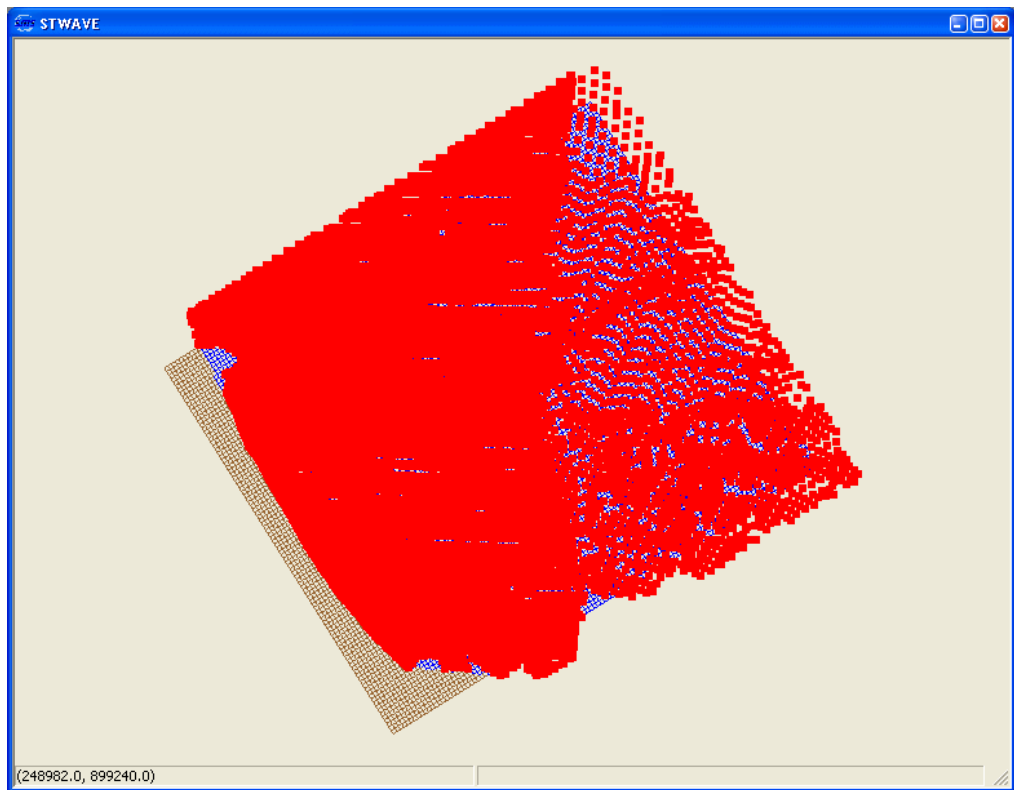


Figure 24-3. Bathymetry points over STWAVE grid.

24.5 Changing Display Options

It will be easier to see the bathymetry values associated with this grid if some of the display options are changed. To see the contours:

1. Select *Display | Display Options* or click the *Display Options*  macro, or right click “Cartesian Grid Data” in the project explorer and select *Display Options*.
2. In the *Cartesian Grid* tab, turn on the *Contours* option and turn off the *Cells* option.
3. In the *Contour Options* tab, change the *Contour Method* to *Color Fill* and change the *Number of Contours* to 12.
4. Click the *OK* button to accept these changes.
5. Deselect the “STW_bath” scatter set in the project explorer to turn off the display of the scatter points.

The display will show contours of depth values assigned to the grid when it was created, as shown in Figure 24-4.

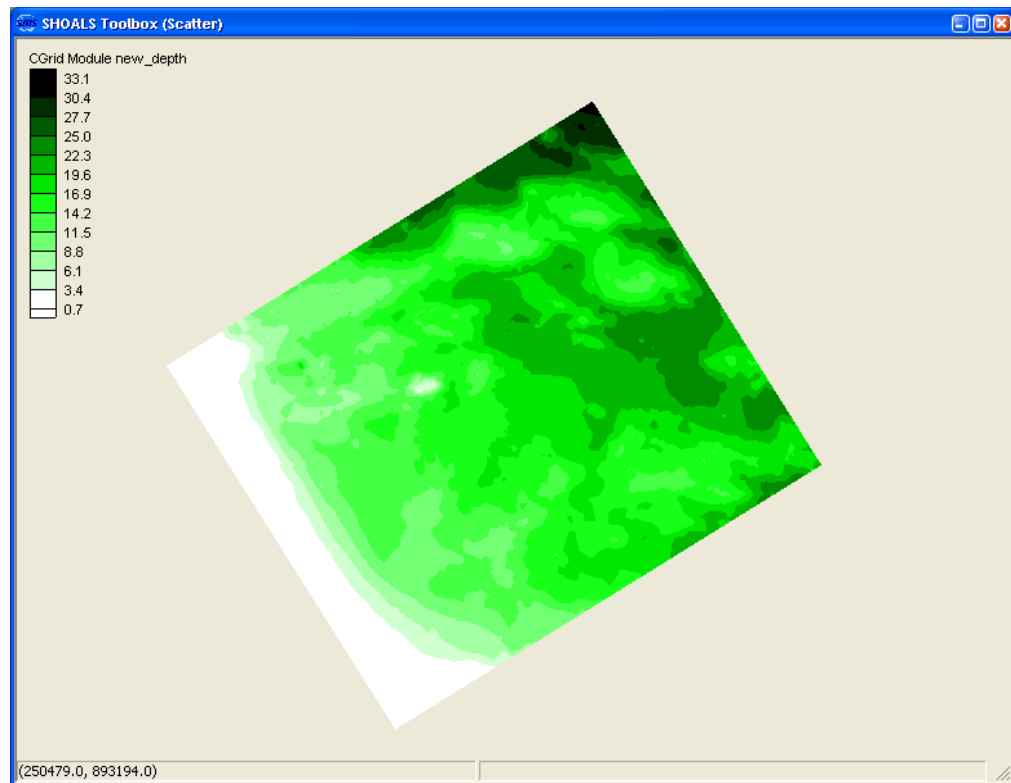


Figure 24-4. Contours of depth.

24.6 Opening Coastline Data

Now that the STWAVE grid exists, the GENESIS grid can be created. Existing coastline data will be opened to define the simulation starting point. For this case, the initial coastline is a survey from 1974 as a series of x/y locations in the file named “1978_sho.XY”. The easiest way to get the coastline into SMS is to first convert it to a coastline file. To do this:

1. Select *File | View Data File* and open the file “1978_sho.XY”.
2. Choose a text editor such as Notepad or WordPad and click *OK*.
3. Add three new lines at the beginning of the file. The first line identifies the file as a coastline file. The second line has the number of coastlines in the file (one). The third line tells the number of points in the coastline (3449 in this case).
4. The first few lines of the file should look like Figure 24-5.

5. Save the file and close the text editor.

```
COAST
1
3449 0
250861.7831 895325.8827
250861.9964 895324.3587
250862.2403 895322.8347
```

Figure 24-5. Modified coastline file.

For more information on coastline files, see the *SMS Online Help*. The data in the file can now be opened as a coastline for use with GENESIS. To open this coastline:

1. Select *File | Open* and open the file “1978_sho.XY”.

A coastline file is officially part of CGWAVE and ADCIRC only, so it must be imported into one of these coverage types. Once imported, the type can be changed to allow working with other models. To change the coverage to GENESIS:

1. Right click “default coverage” under the *Map Data* folder and select *Type->GENESIS*.

An arc defining the coastline should be in the shallow part of the grid, as shown in Figure 24-6.

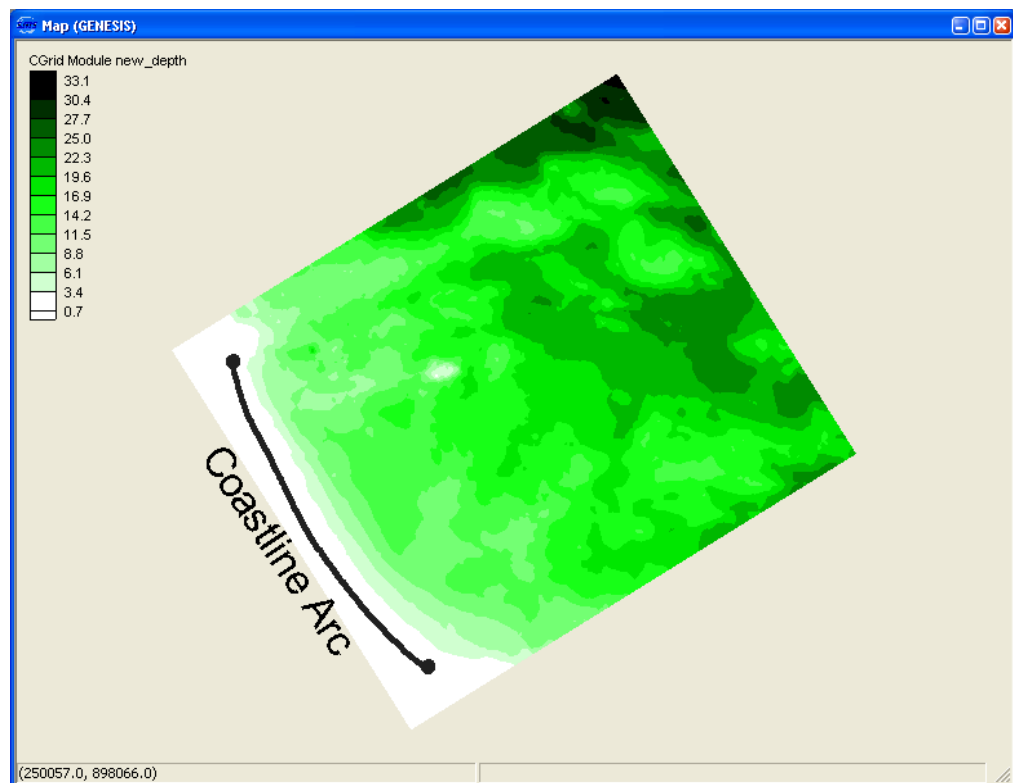


Figure 24-6. The initial coastline for GENESIS.

After the arc exists and the coverage is changed to GENESIS, the arc can be assigned as an initial coastline. To assign the arc as the coastline:

1. Choose the Select Feature Arc  tool from the Toolbox.
2. Double-click on the coastline arc.
3. Change the *Arc Options* to *Initial Coastline*.
4. Click *OK* to change this arc's attribute.

24.7 Defining the Genesis Grid Frame

The GENESIS grid should be created near the STWAVE grid shoreline. It is created through a 1D grid frame object in the GENESIS coverage. To define the GENESIS grid frame:

1. Select *Feature Objects | New Grid Frame*.
2. Be sure the *Match 2D Grid* option is turned on.
3. Set the *Start Row* to 9 and the *End Row* to 94.
4. Click *OK* to create the GENESIS grid frame.

The grid frame should look like that in Figure 24-7.

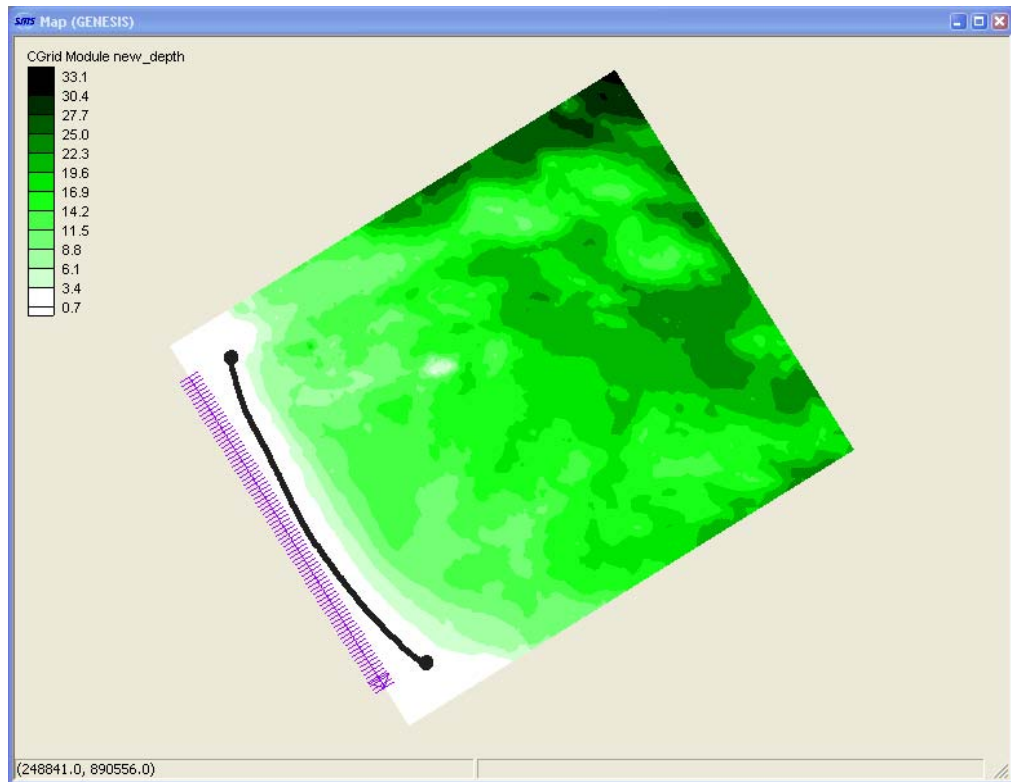


Figure 24-7. The GENESIS grid frame and initial coastline.

24.8 Generating the GENESIS Grid

The genesis grid frame has been defined across most of the CGWAVE grid. With the initial cross section and grid frame defined, the genesis grid can be generated. To do this:

- Select *Feature Objects* | *Map* -> *1D Grid*.

This command creates the actual grid object that will be used with the GENESIS model and samples the coastline arc with the grid spacing. A *Genesis Data* folder and a *Genesis Grid* will appear in the project explorer once it is created. It would be a good idea to save the feature objects data for later and then close it from SMS to get it out of the way. To close the feature objects data:

1. Select *File* | *Save As*. Change the *Save as type* to *Map Files (*.map)*.
2. Save the file as "InitialCoastline.map".
3. Select *Feature Objects* | *Delete* and click *Yes* to the prompt.

The feature objects will not longer be visible. Only the GENESIS data will be displayed above the CGWAVE grid.

24.9 Opening Wave Data

Now that the grid has been initialized, wave data can be applied to this simulation. A wave file contains wave height, period, and direction information. These data could include primary and secondary wave components or just primary components. The wave data for this simulation has primary wave components at each hour for ten years, from 1990 through 1999. To apply these wave data:

1. Select *File | Open* and open the file “Sta_052.nw”.
2. In the *Open File Format* dialog select the *Use Import Wizard* option and click the *OK* button.
3. The first row is a header row so turn on the *Heading row* option and click the *Next* button to get to *Step 2* of the *File Import Wizard*.
4. In the *SMS data type* section, select the *Wind, Wave, Water level* option.
5. Set the *Type* in the five columns to *Date*, *Time*, *Primary Height*, *Primary Period*, and *Primary Direction*, respectively, then click the *Finish* button.

After a few moments the wave data will be opened into SMS and the *Ocean Conditions* dialog will appear. Keep this dialog open through the next few sections of this workshop.

24.10 Transforming Wave Data

Wave information is incorporated into the model at the edge of the STWAVE grid. The waves are usually measured at some offshore location that does not coincide with the grid’s edge. The program WAVTRAN developed by ERDC translates the wave information onto the edge of the STWAVE grid using WIS Phase III (WISPH3) spectral transformations. To run the WISPH3 filter:

1. Change to the *Station Information* tab.
2. The average water depth at the buoy where these waves were measured was 63 meters, so enter this value in the *Station depth* field.
3. The average depth along the ocean side of the STWAVE grid is 25 meters so enter this value in the *Offshore depth* field.
4. Change back to the *Filters* tab.
5. Set the *Filter* option to the *Wave transform (WISPH3)* filter.
6. Click the *Apply* button to transform the waves.

The WISPH3 transformation will take a few moments to run during which time SMS will not respond. When WISPH3 is finished, the Apply button just clicked will become disabled and the spreadsheet list of wave events will update. This list will contain some blue rows, indicating that WISPH3 flagged those events as calm with respect to the shoreline angle. To display only active wave events in the list:

- Turn off the *Display calm events* option.

24.11 Changing the Angle Convention

SMS displays a selected wave event at the top left of the *Ocean Conditions* dialog. Within this window, the North arrow, shoreline and wave direction are all indicated. Any event can be selected from the *Primary Waves* spreadsheet. To display the wave event at midnight January 1, 1990:

- Click on the first row of the Primary Waves spreadsheet. The wave picture will look like Figure 24-8.

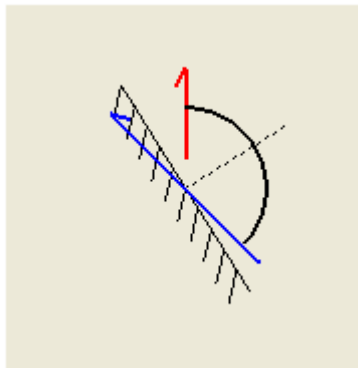


Figure 24-8. The wave event at 0:00 hours on 1 January 1990.

The angle convention determines how wave directions are measured. Changing the convention does not change wave events; it simply changes the way they are viewed. When wave events are opened from a file, SMS assumes the angles to be in *Meteorological* convention, measured clockwise from north to the wave's tail. It is sometimes easier to visualize waves in *Shore Normal* (local polar) convention, measured counter-clockwise from the shore's normal vector to the wave's tail. For this convention, a wave with a direction of zero is traveling exactly toward shore. To switch the angle convention:

- Change the *Angle Convention* drop down box to *Shore normal*. SMS will update the wave directions using this convention.

24.12 Creating Event Bins

Even though WISPH3 removes some wave events, there are still hundreds of events that must be considered in the analysis. Instead of modeling each individual event, representative bins are created by defining wave height, period and direction ranges. Each wave event is placed into the bin that most closely describes it. The average values for each bin define the “characteristic” wave of the bin and only the set of characteristic wave events are modeled.

In SMS, bins are always defined with *Shore Normal* orientation. This is the case no matter the setting of the *Angle Convention* option. To define the bins for this model:

1. Select the *Define Bins* tab.
2. Set the bin ranges for Height, Period and Angle as shown in Table 24-1.
3. Click the *Apply* button to apply the new wave bin values. SMS will put each wave event in the corresponding bin.

Table 24-1. The values that define bins.

Parameter	Bin Values									
Height	0.0	0.5	1.0	1.5	2.0	3.0	4.0	6.0	8.0	
Period	0.0	5.0	7.0	9.0	11.0	13.0	15.0	17.0	23.0	
Direction	-90.0	-30.0	-15.0	0.0	15.0	30.0	90.0			

The permutations page lists each bin along with its number of events and average wave event. Only those bins containing at least one event will be shown. At the top of this table SMS displays the number of calm events and the number of unbinned events (events not in a bin), if any. To view this permutation information:

- Select the *Permutations* tab. There should be 12266 calm events and no unbinned events followed by information for each bin.

SMS can display a bar plot showing the number of wave events inside each bin. To display this plot:

- Turn on the *Display plot* option. The dialog should be similar to Figure 24-9.

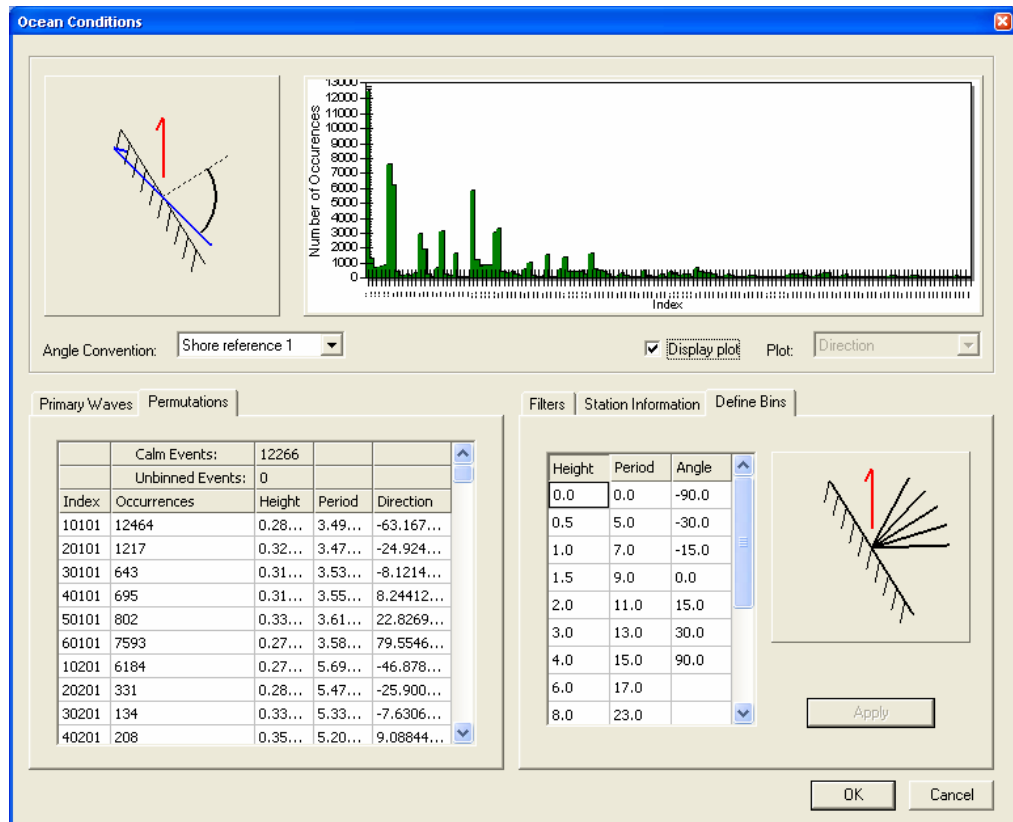


Figure 24-9. Wave bin values and frequency bar plot.

Notice that the number of wave events is greatly biased toward a few bins, and most bins have very few events in them. For this simulation, the existing bins will suffice, but consideration could be taken to produce a more even distribution of the wave events throughout the bins.

24.13 Applying Additional Wave Data Filters

In addition to transforming the wave data, other filters may be applied. These filters help to refine the exact range of wave, period and/or direction to allow within the simulation. To filter the period:

1. Switch to the *Filters* page.
2. Change the *Filter* type to *Wave period range*.
3. Leave the *Calm events* flag as -1.0.
4. Change the *Max* value to 10.0 and press the *Apply* button.

After a few moments, the plot will slightly change and the number of *Calm Events* at the top of the *Permutations* page will be reported as 14857. To see which events were taken out by this filter:

1. Before continuing, turn off the *Display Plot* option. The number of wave events is so high that it would take a lot of time to plot them.
2. Change back to the *Primary Waves* page.
3. Turn on the *Display calm events* option.
4. Scroll through the list of events to find some events with a period of -1.0.

Any wave event with a negative period is a calm event. A period of -1.0 was set as such when the *Wave period range* filter was applied. For example, the event at hour 20:00 on 17 January 1990 should have this value for its period. For this simulation, all wave events will be considered so the filter needs to be undone. To do this:

1. Click the *Undo* button. Be sure to only click this button once or you will also undo the previous WISPH3 computations. It may take a few moments for SMS to undo the filter, after which the *Max* value for *Wave period range* will return to 16.67. (If you accidentally undo both filters you can redo the WISPH3 filter by clicking the *Redo* button).
2. You are now done with the *Ocean Conditions* dialog. Click the *OK* button to continue working on this simulation.

24.14 Generating STWAVE Spectra

The wave event bins are used to generate wave spectra for use with STWAVE. To import this data into STWAVE:

1. Switch to the *Cartesian Grid*  module.
2. Select *STWAVE | Spectral Energy*.
3. Click the *Create Grid* button to define a spectral energy grid. Change the *Minimum frequency* to 0.06 Hz then click OK.
4. Click the *Generate Spectra* button. From the bottom of the dialog, click the *Import From Genesis* button. The table will be filled with the average value from each of the 138 wave bins.
5. Change the water depth to 25.0 for all spectrum and leave the *Gamma* and *nn* values at 3.3 and 4, respectively, and click the *Generate* button. The table will be filled with energy spectra. When one is selected, a plot displays the result of the spectrum.

6. Click the *Done* button to close the *Spectral Energy* dialog.

After the wave spectra have been defined, you need to verify that they will all be used in the simulation. Individual spectra can be enabled or disabled for a given STWAVE simulation. To make sure all spectra will be used in this simulation:

1. Select *STWAVE | Model Control*.
2. Click the *Select Input Spectra* button. Be sure the check mark next to the “New_Spectral_Grid” is turned on and click the *OK* button. This verifies that all the spectral data will be used in the STWAVE simulation.
3. Click the *OK* button to close the *STWAVE Model Control* dialog.

24.15Running STWAVE

With the energy spectra defined, STWAVE can now be run. Each energy spectrum will be applied to the boundary of the grid. STWAVE will track the wave energy through each grid cell. To run STWAVE:

1. Choose *File | Save STWAVE*.
2. Enter the name “mass.sim” and click the *Save* button.
3. Choose *STWAVE | Run STWAVE*. The path to stwave.exe should be shown. If the message “<Not Found>” appears, click the *File Browser*  icon and find the executable file.
4. Click the *OK* button to run *STWAVE*.

As the model runs, the dialog shown in Figure 24-10 will keep track of its progress and estimate the time remaining.

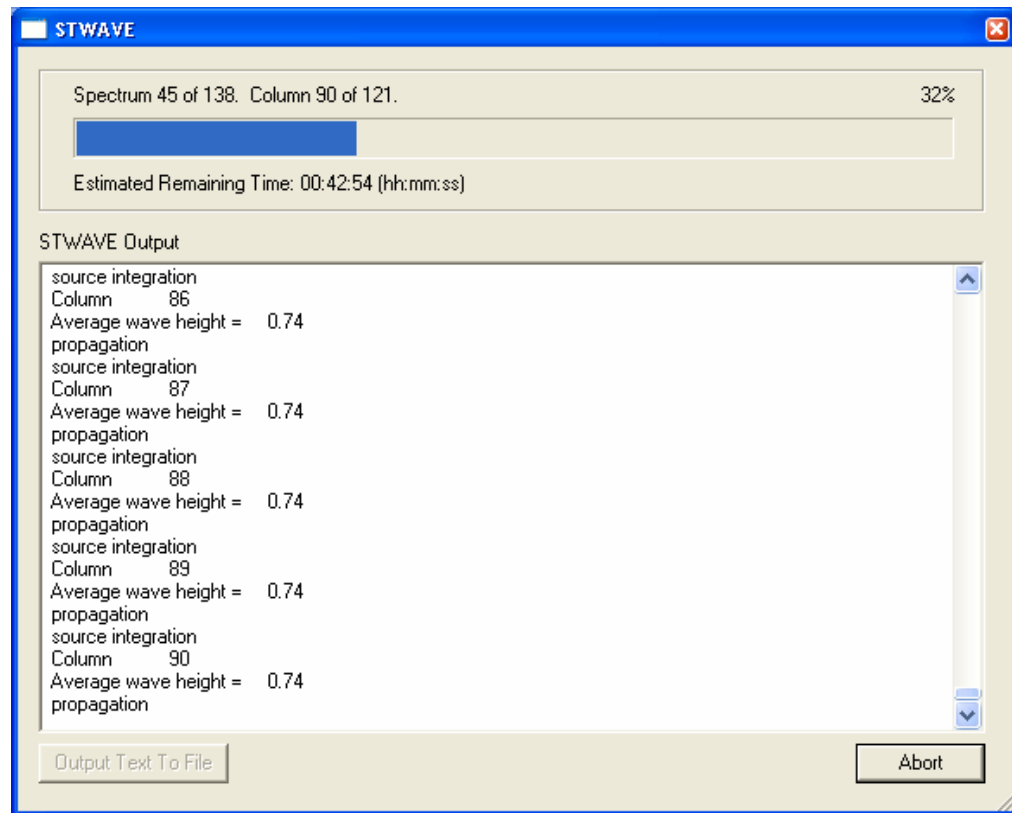


Figure 24-10. The STWAVE progress dialog.

This simulation may take about an hour to run, depending on the speed of your computer. After it has finished, the STWAVE solution should be opened into SMS. To open the solution:

1. When the STWAVE model has finished running, the *Abort* button changes to *Exit*. Click the *Exit* button.
2. Choose *File | Open* and open the file mass.wav.

This STWAVE file type will often be associated with a media player because of its file extension (.wav). When this solution is opened, SMS will create one vector and three scalar datasets. The scalar datasets are wave height, period and direction (in degrees). The vector dataset shows the wave direction as a vector quantity. Each “time step” of the simulation is the solution for the average wave event from the specified bin.

24.16 Assigning Observation Stations

The results from STWAVE will be used as input to GENESIS. Instead of using the entire grid of results, GENESIS uses the values at cells called *monitoring stations* or *observation stations*. There should be one observation station in each row of the

STWAVE grid within the GENESIS grid. For this simulation, observation stations will be created at a constant depth of 5.5 meters. To define the observation stations:

1. Select *STWAVE* | *GENESIS Observation Stations*.
2. Choose to create an observation station at a depth of 5.5 meters and click the *OK* button. SMS will create observation stations matching the 5.5-meter depth contour. This contour also matches the shape of the initial coastline. To better view these cells:

1. Select *Display* | *Display Options* or click the *Display Options*  macro.
2. In the *Cartesian Grid* tab, turn off the *Contours* option and turn on the *Cells* option.
3. Be sure the *Genesis Observation Symbol* option is on. Change its color to red and symbol to a square, size 5.
4. In the *One Dimensional Grid* tab, change the thickness of the *Initial Coastline* option to 3.
5. Click the *OK* button to close the *Display Options* dialog. The display should look like Figure 24-11.

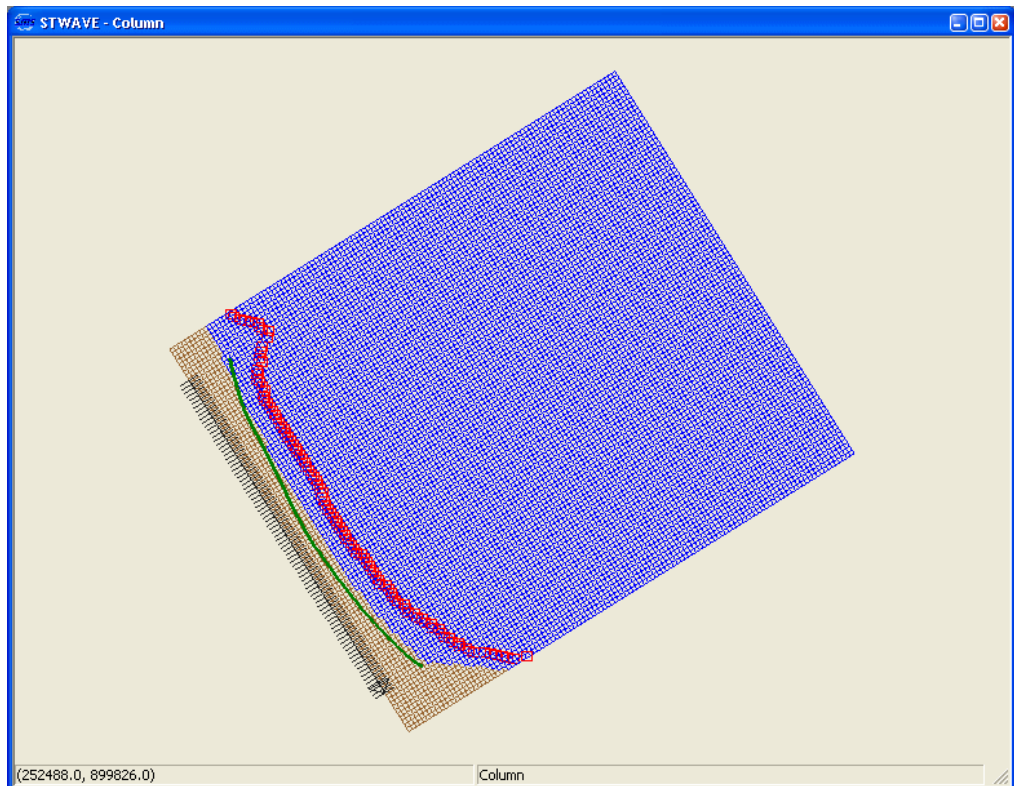


Figure 24-11. The STWAVE grid with observation stations assigned.

24.17 Setting up GENESIS Time Data

The GENESIS simulation time must be set up. To set up the time data:

1. Switch to the *1-D Grid*  module.
2. Select *Genesis | Model Control*.
3. In the *Model Setup* tab, set the *Start Date* to January 1, 1990 and the *End Date* to December 31, 1999.
4. Leave the *Time Step* at 1.0 hour but change the *Recording Time Step* to 24.0 hours. This will output one time step per day in the final solution.
5. Click the OK button to close the *GENESIS Model Control* dialog.

24.18 Running GENESIS

You are now ready to run the GENESIS simulation. To run Genesis:

1. Select *File | Save New Project*.
2. Save the project as mass.sms.
3. Choose *GENESIS | Run Genesis*. The path to genesis.exe should be shown. If the message “<Not Found>” appears, click the *File Browser*  icon and find the executable file.
4. Click the *OK* button to run *GENESIS*.
5. When GENESIS finishes, the *Abort* button changes to *Exit*. Click the *Exit* button.

When GENESIS is finished, it writes the coastline information back into the simulation file. This simulation should be reopened to read the solution. To open the GENESIS simulation file:

- Select *File | Open* and open the file mass.gen.

24.19 Displaying Shorelines

With a solution open, the shoreline from a date can be displayed. To set up the display options for visualizing multiple shorelines:

1. Select *Display | Display Options* or click the *Display Options*  macro.
2. On the *One Dimensional Grid* page, turn on the *Current Coastline*, *Minimum Coastline* and *Maximum Coastline* options.
3. Click the *OK* button.

The change between shorelines can be very small. You may need to zoom into part of the coastline to see such change, but there should be a minimum and maximum coastline, as well as the initial and active coastline. To see this:

- Zoom into a section of the coastline as shown in Figure 24-12.



Figure 24-12. Minimum, maximum and active coastlines.

The Data Tree window displays the coastline at each date/time that the solution was generated. In this case, there should be a coastline at hour zero for each day between 1990 and 1999. To display the coastline at a different time:

- Select a date from the data tree window, as shown in Figure 24-13.

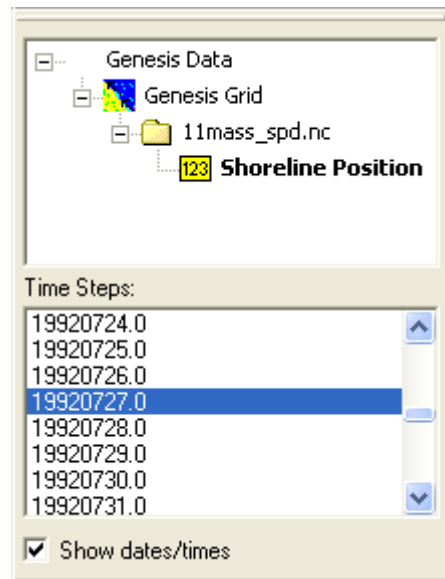


Figure 24-13. Dates in the data tree window.